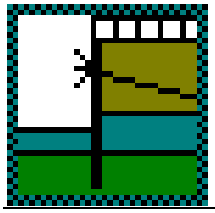
Analysis and design of sheet pile walls, soldier pile walls, in-situ concrete walls, CMG walls and combined sheet pile walls

GGU-RETAIN

VERSION 8



Last revision: October 2015

Technical implementation and sales: Civilserve GmbH, Steinfeld

Contents:

List of Figures:

1 Preface

GGU-RETAIN allows the analysis of sheet pile walls, soldier pile walls and in-situ concrete walls (bored pile walls, diaphragm walls and contiguous pile walls) as well as CMG (cut-mix-grout) walls and combined sheet pile walls.

The application is essentially based on the Recommendations of the Working Group for Excavations (*German: EAB*) and the Recommendations of the Committee for Waterfront Structures (*EAU 2012*). Virtually all the cases considered by the Working Groups are included as examples. A special feature of the application is automatically finding the earth-pressure redistribution values suggested by the EAB. For analysis and design both the ***global safety factor concept*** to DIN 1054 (old) and the ***partial safety factor concept*** to EC 7 may be taken into consideration. Details of special features can be found in Section 2 of this manual.

The application is designed to allow simple data input or modification. The input is immediately shown on the screen, giving you optimum control over what you are doing. Reading of the manual can mostly be dispensed with, because

"**?**" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "**?**" button (see also Section 9.1).

Graphics output supports the true-type fonts supplied with WINDOWS, so that excellent layout is guaranteed. Colour output and graphics (e.g. files in formats BMP, JPG, PSP, TIF, etc.) are supported. DXF files can also be imported by means of the integrated Mini-CAD module (see the "**Mini-CAD**" manual).

The application has been tested on numerous literature examples and in practice. It is now used by a large number of civil engineering firms, construction companies and university institutes, and has successfully completed well over 1000 static tests. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Capabilities

GGU-RETAIN has the following characteristics and maximum capabilities

- Up to 50 soil layers
- Up to 20 berms on the active earth pressure side
- Up to 20 berms on the passive earth pressure side
- Analysis with active and increased active earth pressure and with at-rest earth pressure
- Coefficients of active earth pressure pursuant to DIN 4085 and user-defined values
- Coefficients of passive earth pressure pursuant to DIN 4085, Streck, Caquot/Kerisel and user-defined values

- Classical water pressure approach for impermeable retaining walls and, alternatively, by means of flow conduits parallel to the wall in accordance with potential theory (analysed using finite-element methods). Inconsistencies in subsurface hydraulics apparent when using the classical approach are avoided when using flow conduits, and varying permeabilities are correctly considered as far as the approach allows. User-defined potentials can be applied at any point along the flow conduit. This allows correct, problem-free consideration of systems with several groundwater storeys and/or aquifuges, or with artesian conditions.
- Optional consideration of hydraulic gradients on the active and passive earth pressure sides
- Verification of safety against deep-seated failure with optimisation of anchor lengths
- Analysis of hydraulic heave 'classical' or after Aulbach/Ziegler
- Verification of buoyancy safety
- Verification of base heave safety
- Verification of pull-out resistance of driven steel tubular piles and injection piles
- Verification of sum H
- Analysis of mobilised passive earth pressure
- Analysis of vertical capacity
- Convenient interface to the stability analysis application, **GGU-STABILITY**, for quick determination of safety factor against general failure
- Up to 50 additional earth pressure distributions
- Up to 20 area loads at any depth
- Structural analysis of the retaining wall by means of a two-dimensional rod construction module based on finite-element methods. In contrast to many other retaining wall analysis applications, the influence of inclined anchors or struts and their interactions are thus directly considered in the analysis approach. Analysis can even be performed using 2nd order theory, making the generally onerous buckling length investigation of struts and retaining wall unnecessary. In addition, it is possible to consider loads on the struts (e.g. for auxiliary bridges additionally acting as struts) during analysis.
- Buckling analysis to DIN EN 1993-1-1 using 2nd order theory
- Elastic subgrade reaction at the wall toe with user-defined values. Option for automatic selection of an elastic modulus of subgrade reaction profile consistent with the specified passive earth pressure distribution
- Specification of up to 5 displacement boundary conditions (rotation, displacement in x or y directions) at any location
- Specification of up to 5 action boundary conditions (moment, shear force and normal force) at any location
- 20 anchor and strut sets; in defining the anchors and struts, axial stiffness and bending stiffness can be specified, so that, for example, passive anchors can be incorporated.
- Consideration of prestressing of anchors and struts possible
- Definition of up to 20 additional potentials by the calculation of flow conduits to the left and right of the wall for complex groundwater conditions
- Expandable database of default sections for soldier piles, sheet piles, combined sheet piles and tubular sections with which automatic design, including automatic searches for the optimum section, can be carried out.
- Reinforced concrete design to EC 2 for circular and rectangular cross-sections
- Modelling corrosion for all sections

- Design of horizontal lagging for soldier pile walls
- Automatic computation of section weights for analysis of sum V
- Variable bending resistance of the retaining wall
- Automatic computations of earth pressure redistributions according to the EAB Recommendations
- Other earth pressure distribution options:
  - no redistribution
  - rectangle
  - 2 rectangles
  - triangle (maximum can be optionally placed at top, middle or bottom)
  - trapezoidal
  - quadrilateral with maximum at anchor locations or any other location
  - user-defined polygon
  - EAU 2012
- Soil pressure can be redistributed either to the wall toe or to the transition point.
- Passive earth pressure can be superimposed or located in front of the wall.
- Calculation of the transition point with or without porewater pressure
- Structural analysis can be performed in 4 different ways:
  - section length determined automatically after entering degree of fixation for the wall toe
  - determination of degree of fixation for fixed section length
  - section length automatically determined with elastic bedding of wall toe
  - bedding of wall toe with fixed section length
- Following computation of the system, earth pressure, porewater pressure, moment, shear, normal force and bending line are displayed on the screen. The visualisation can be varied within wide limits. For example, the distribution of the modulus of subgrade reaction as well as the distributions of the potential and the gradient, etc. can also be displayed.
- For soldier pile walls it is necessary to demonstrate the equilibrium of horizontal forces below the excavation level. This is carried out by the application. If necessary, the section length is automatically increased.
- At anchor points, existing predeformations can be defined as boundary conditions.
- Previously calculated datasets can be combined to investigate the additive deformation of individual advancing and retreating states. A Mohr's envelope showing the distribution of moments, shear and normal forces can also be depicted.
- Legends can be displayed on screen showing soil properties and general computation data. Thus, virtually all the raw data necessary for the computation and the results are shown on screen.
- **GGU-RETAIN**'s user interface is based on WYSIWYG (***What You See Is What You Get***), which means that what you see on the screen is virtually identical with what is printed. It also means that you can print out what you see on the screen at any point during the analysis.
- The use of true-type fonts guarantees excellent layout.
- Colour presentation of virtually all system geometries. Colours can be freely determined by the user. In particular, soil strata can also be coloured according to the German DIN 4022 conventions.
- Zoom function
- **Mini-CAD** system (additional text, lines, rectangles, circles, graphics, etc).

- **GGU-RETAIN** includes virtually all the examples contained in the Krupp Hoesch Stahl "Piling Handbook" ("Spundwand-Handbuch") and in Weißenbach's "Excavations" III (*"Baugruben III"*, 1977) as datasets.
- By clicking the "**Copy/print area**" icon on the toolbar you can copy any part of the graphics to the clipboard, save it as an EMF file (Enhanced Metafile Format) or print it directly on your printer. Using the "**Mini-CAD toolbar**" or "**Header toolbar**" modules, you can insert EMF files into your graphics. Thus, the results of a slope stability analysis or of a grading analysis, for example, can easily be imported into the current graphics.

3 Licence protection and installation

In order to guarantee a high degree of quality, a hardware-based copy protection system is used for the **GGU-RETAIN** program.

The GGU software protected by the ***CodeMeter*** copy protection system is only available in conjunction with the ***CodeMeter stick*** copy protection component (hardware for connection to the PC, "***CM stick***"). Because of the way the system is configured, the protected software can only be operated with the corresponding CM stick. This creates a fixed link between the software licence and the CM stick copy protection hardware; the licence as such is thus represented by the CM stick. The correct Runtime Kit for the CodeMeter stick must be installed on your PC.

Upon start-up and during running, the **GGU-RETAIN** program checks that a CM stick is connected. If it has been removed, the program can no longer be executed.

For installation of GGU software and the CodeMeter software please refer to the information in the ***Installation notes for GGU Software International***, which are supplied with the program.

4 Language selection

GGU-RETAIN is a multilingual program. The program always starts with the language setting applicable when it was last ended.

The language preferences can be changed at any time in the "**?**" menu, using the menu item "**Language preferences**" (in German: "**Spracheinstellung**", in Spanish: "**Configuración de idioma**").

5 Starting the program

After starting the program, you will see two menus at the top of the window:

- File
- ?

By going to the "**File**" menu, a previously analysed system can be loaded by means of the "**Load**" menu item, or a new one created using "**New**". After clicking the "**New**" menu item a dialog box opens for specifying general preferences for your new system (see Section 8.1.1). You then see nine menus in the menu bar:

- File
- Editor 1
- Editor 2
- System
- Evaluation
- Construction phases
- Graphics preferences
- Page size + margins
- ?

After clicking one of these menus, the so-called menu items roll down, allowing you access to all program functions.

The program works on the principle of ***What you see is what you get***. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the **GGU-RETAIN** screen is not refreshed after every alteration.

If you would like to refresh the screen contents, press either [**F2**] or [**Esc**]. The [**Esc**] key additionally sets the screen presentation back to your current zoom, which has the default value 1.0, corresponding to an A3 format sheet.

6 First steps using worked examples

6.1 Worked example 1: Soldier pile wall

6.1.1 System description

Knowing from experience that having to work one's way through a software manual can be very tiresome, the following sections provides a short description of **GGU-RETAIN**'s main functions, which will quickly enable you to carry out a retaining wall analysis. Details, when needed, will be found in the appropriate chapter of this manual. The following example of a retaining wall is to be analysed:

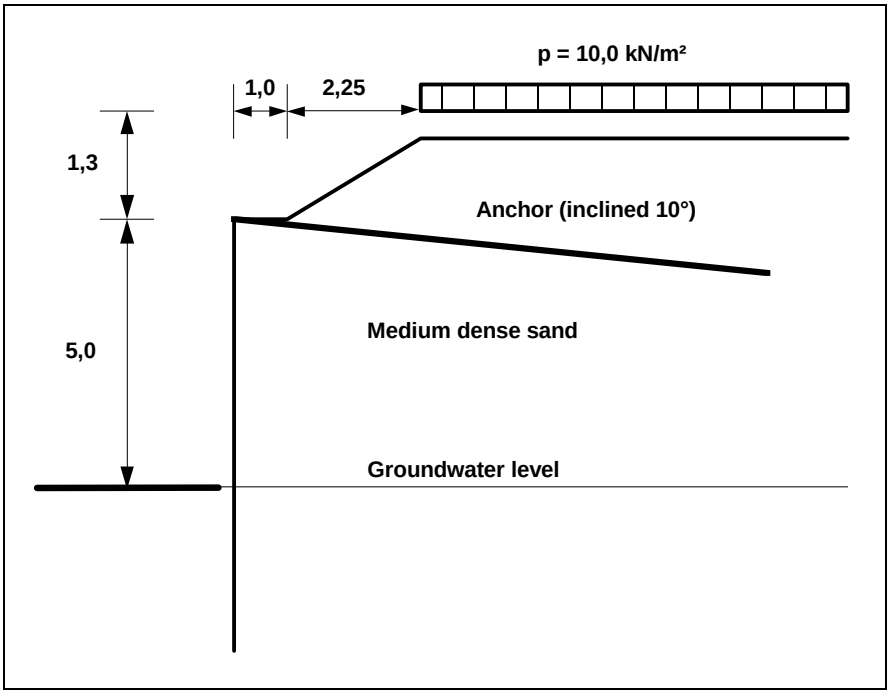
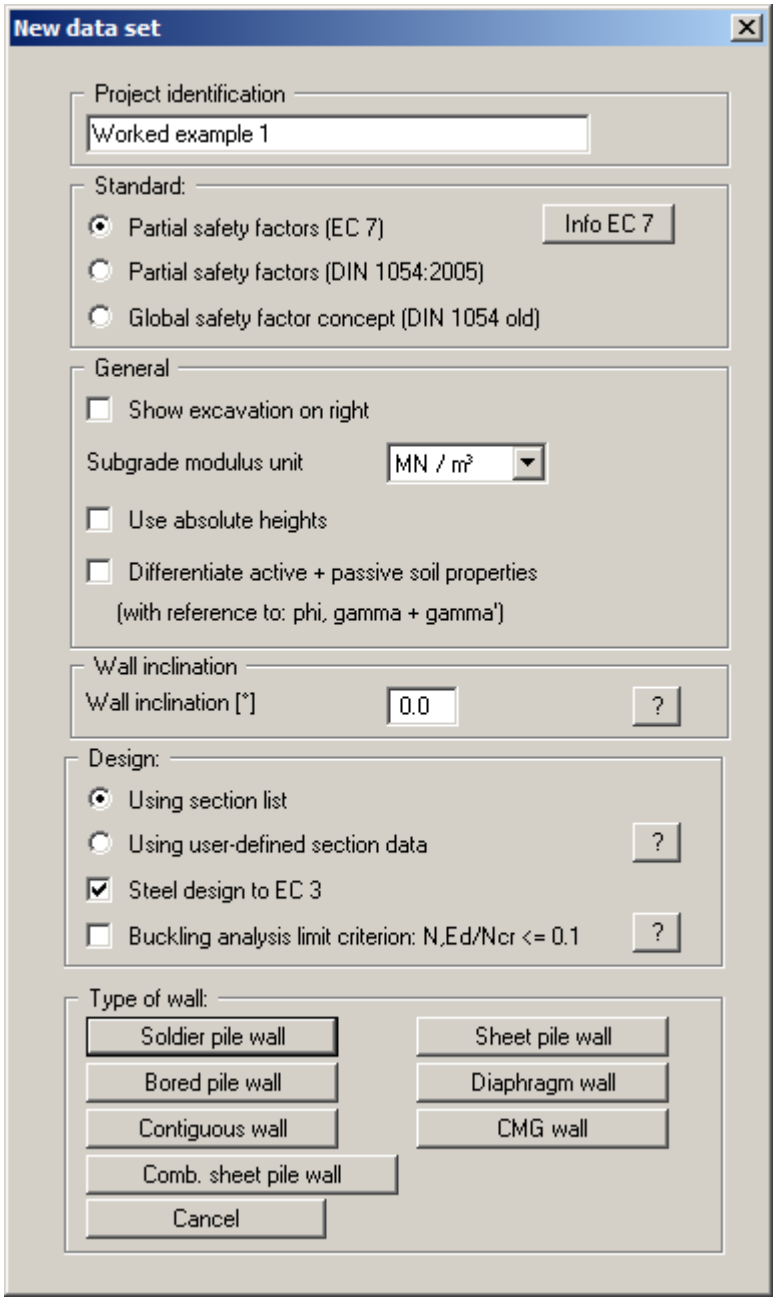


Figure 1 Illustration of worked example 1

The example is a soldier pile wall with an anchor at the top of the wall. The groundwater level coincides with the excavation base. On the active side there is a berm, subjected to a load of 10 kN/m². The soldier pile centres are 2.2 m. "**HR 300**" H-piles are to be used and analysed with a free earth support.

6.1.2 Step 1: Select analysis options

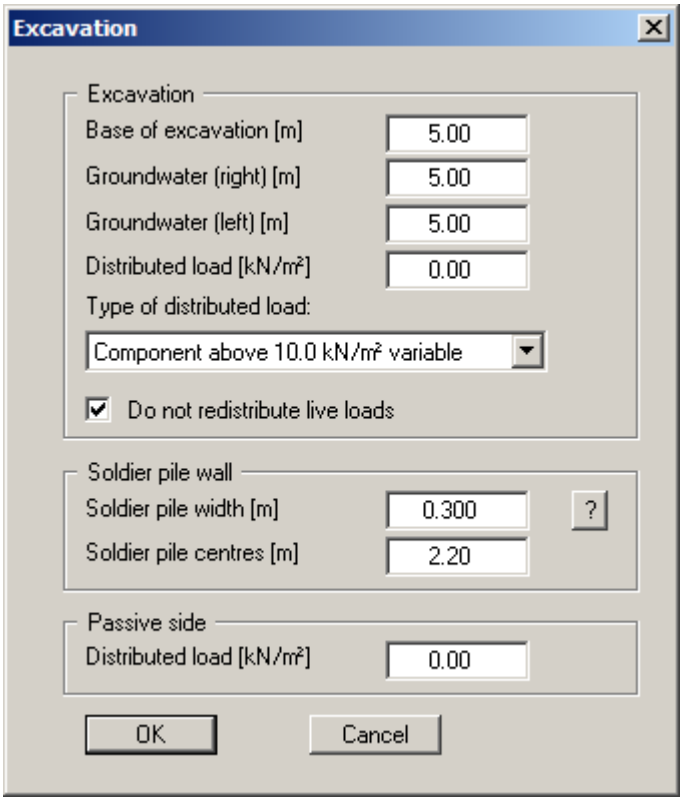
After starting the program the logo is displayed. Select the menu item "**File**/**New**". The following dialog box will appear:



Select the buttons shown in the above dialog box and click "**Soldier pile wall**". A new system is displayed on the screen and the complete menu bar is activated.

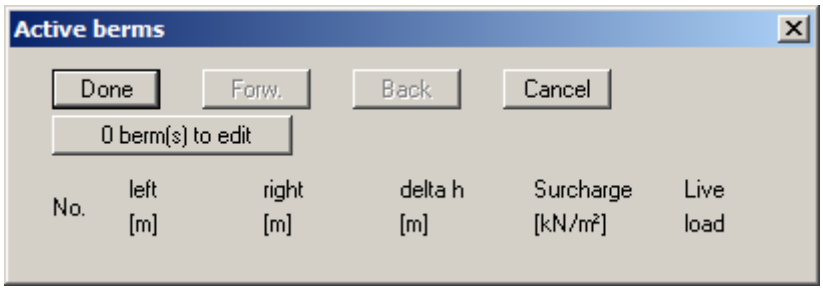
6.1.3 Step 2: Define excavation and retaining wall

From the "**Editor 1**" menu select "**Excavation**". The following dialog box will appear. Enter the figures shown below:

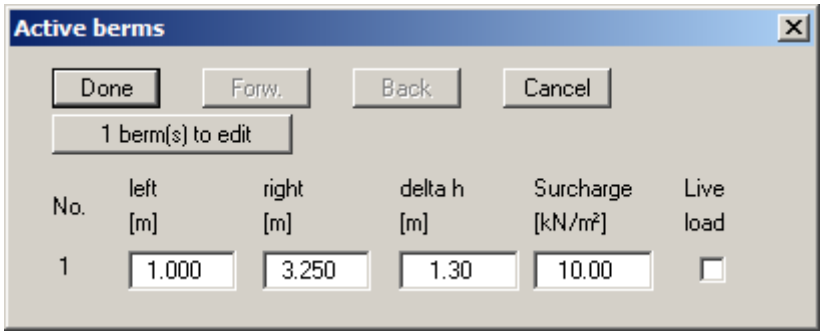


6.1.4 Step 3: Define berm

Go to the "**Editor 1**" menu and select "**Berms (active side)**":

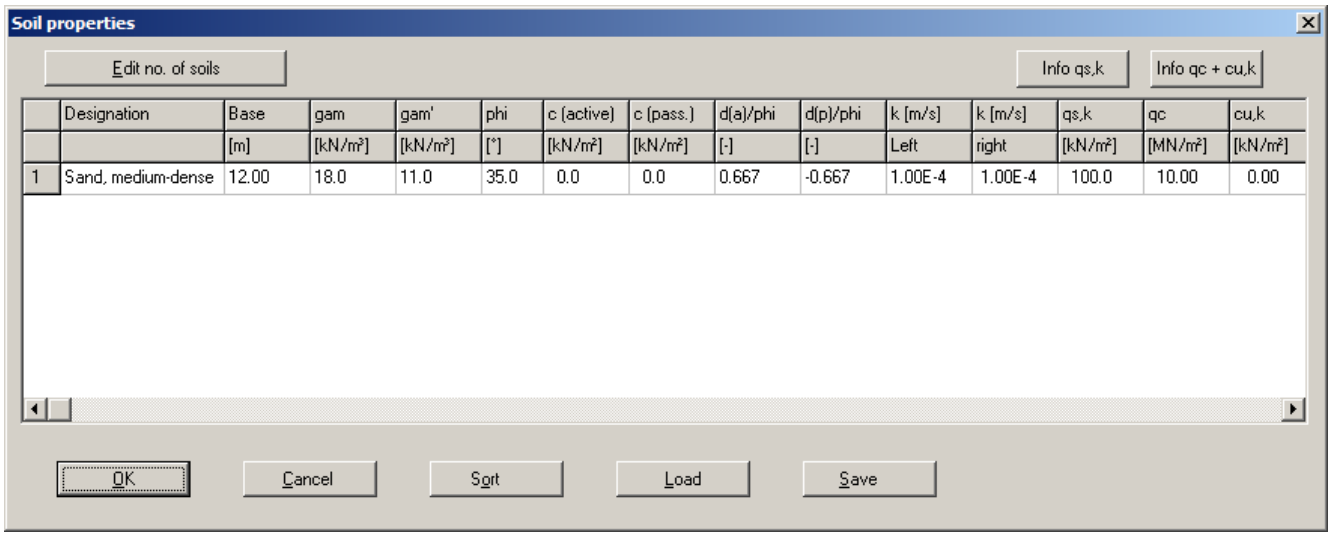


Click "**0 berm(s) to edit**" and enter **1** as the new number of berms. Enter the following values and click "**Done**".



6.1.5 Step 4: Define soils

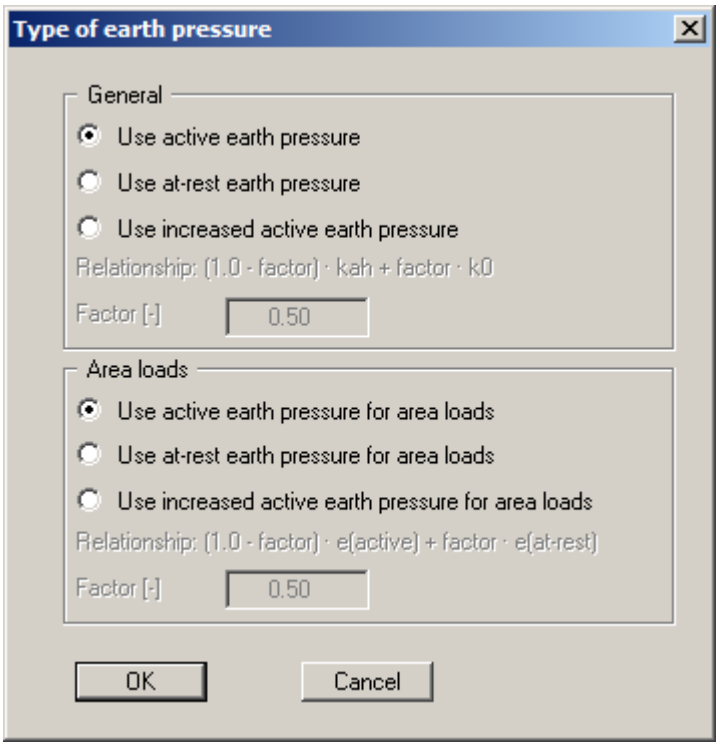
Go to the "**Editor 1**" menu and select "**Soils**".



Enter the values shown in the above dialog box.

6.1.6 Step 5: Define type of earth pressure

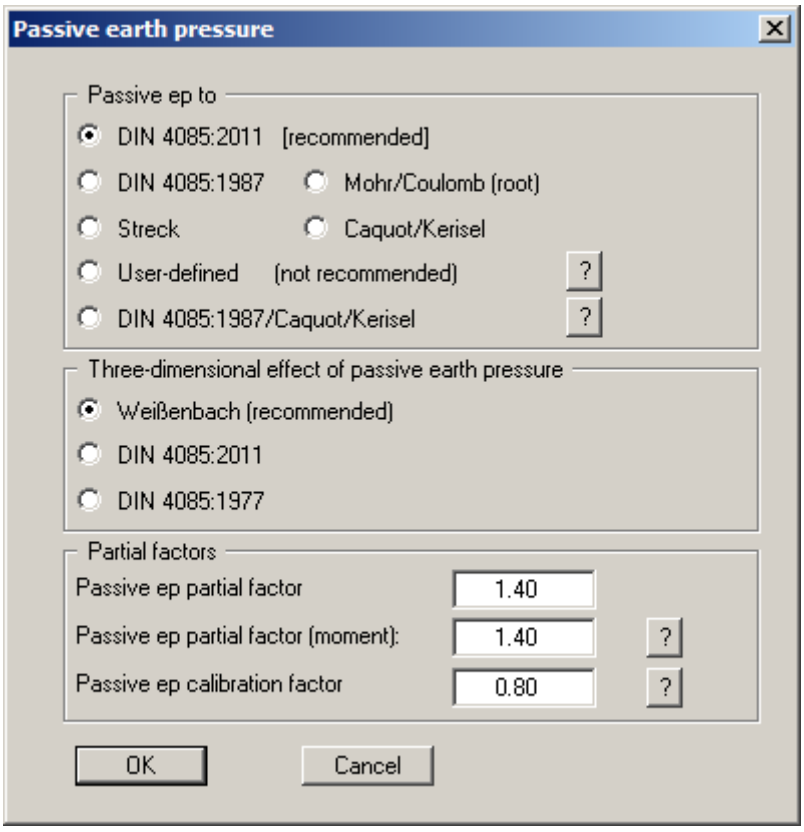
Go to the "**Editor 1**" menu and select "**Type of earth pressure**".



The necessary command buttons are already selected, so you need not change anything. The same applies to the remaining menu items in "**Editor 1**". However, you should click on these items and take a look at them, in order to familiarise yourself with them.

6.1.7 Step 6: Define passive earth pressure

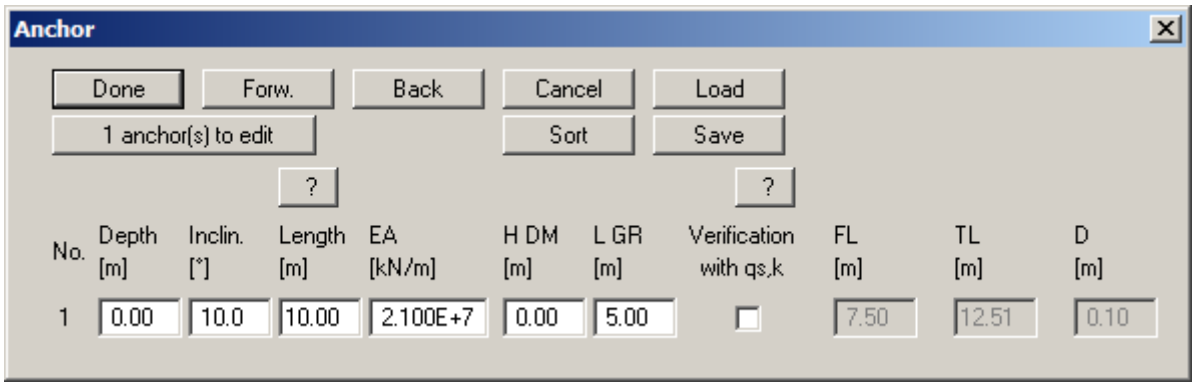
Go to the "**Editor 1**" menu and select "**Passive earth pressure**".



Accept the proposed setting of "**DIN 4085:2011**".

6.1.8 Step 7: Define anchors

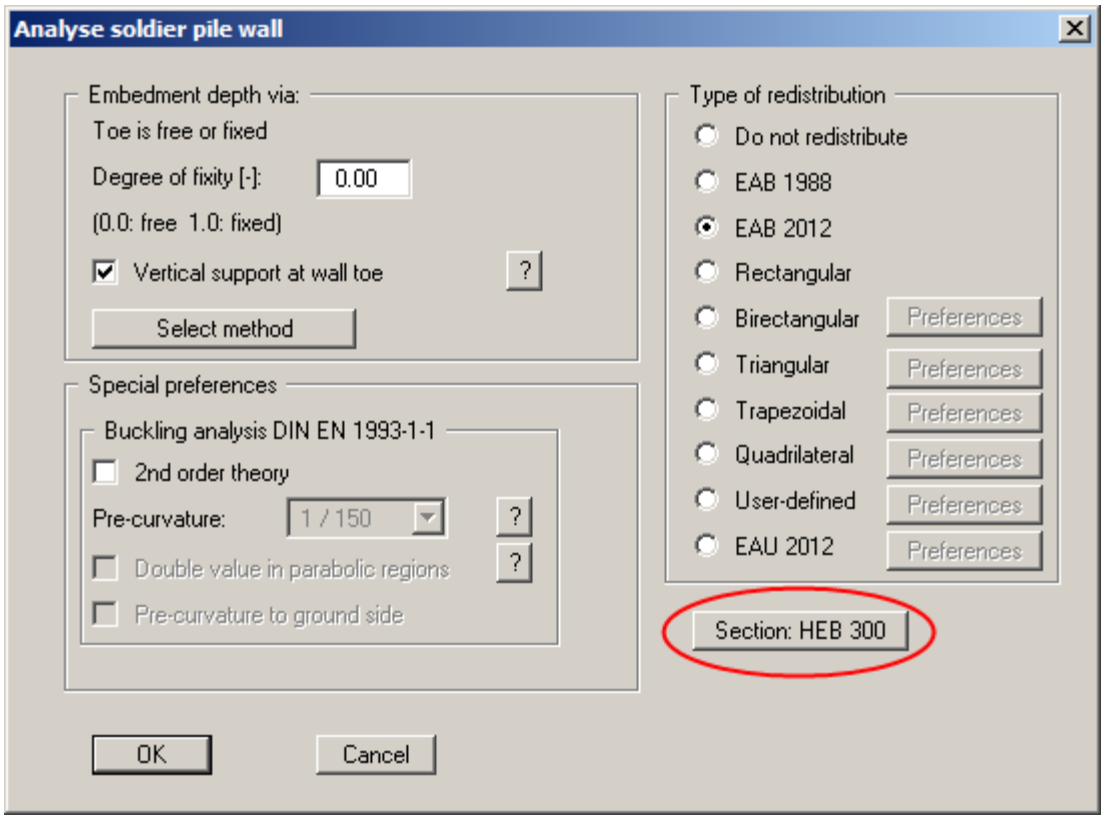
Go to the "**Editor 2**" menu and select "**Anchors**". Now click "**0 anchor(s) to edit**" and set the number to **1**.



Enter the values shown in the above dialog box. Click "**Done**" and you have completed data entry.

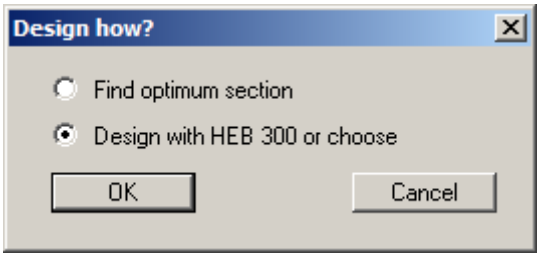
6.1.9 Step 8: Analyse and design the system

Go to the "**System**" menu and select "**Analyse**".

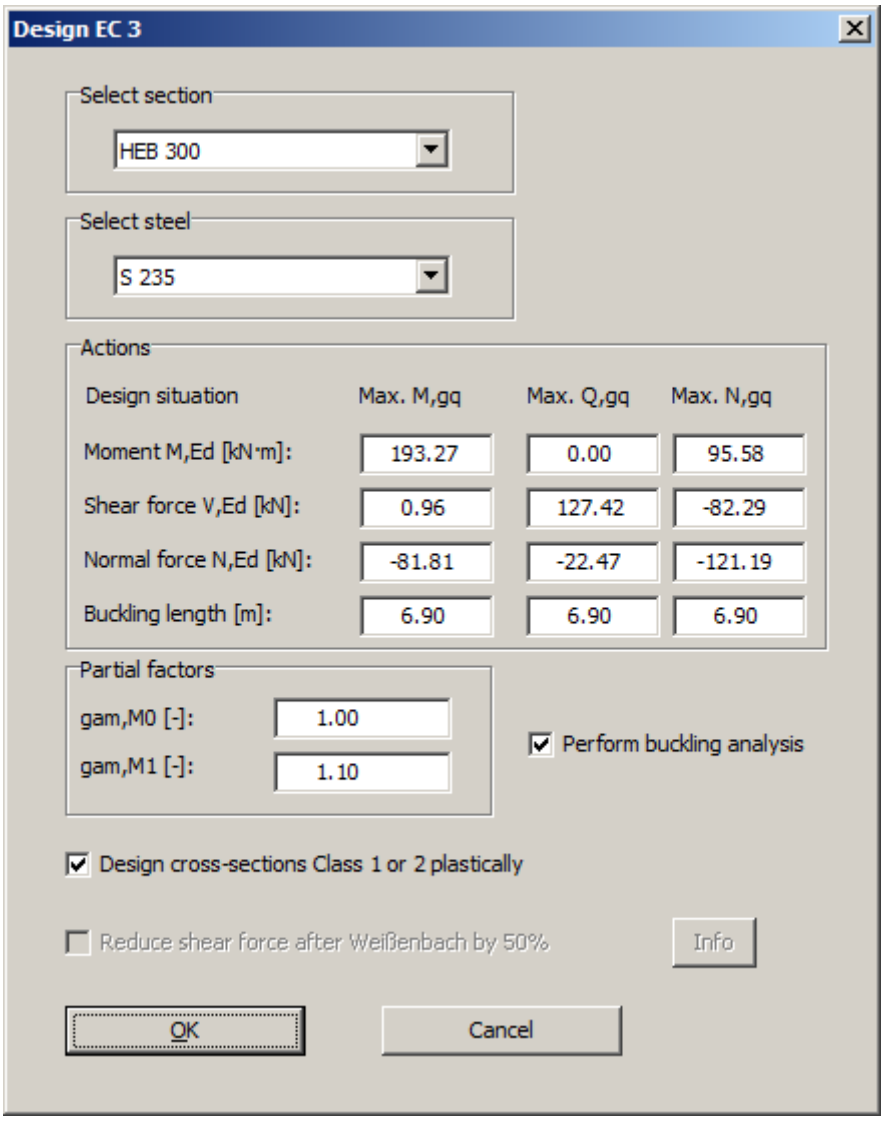


Select a "**Degree of fixity**" = "**0.0**" (= free earth support). Earth pressure redistribution is in accordance with the EAB 2012 (activate the appropriate command button). The required section can be selected using the marked button, which is set to "**Section: HEB 100**" when the program starts. For the example, select "**HEB 300**" from the list.

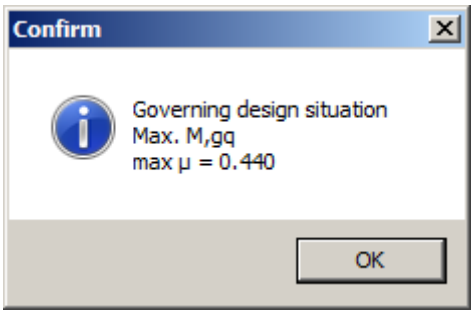
The analysis can then be started by pressing "**OK**". When analysing a soldier pile wall you will see a message stating that a passive earth pressure calibration factor of 0.8 should be adapted for analysis. This can be corrected by the program. Following the analysis you can decide whether or not to carry out design. If you answer the question with "**Yes**" you will see the following dialog box:



For the example, select the "**Design with HEB 300 or choose**" check box and confirm by pressing '**OK**'. The dialog box with the settings for design to EC 3 then opens:

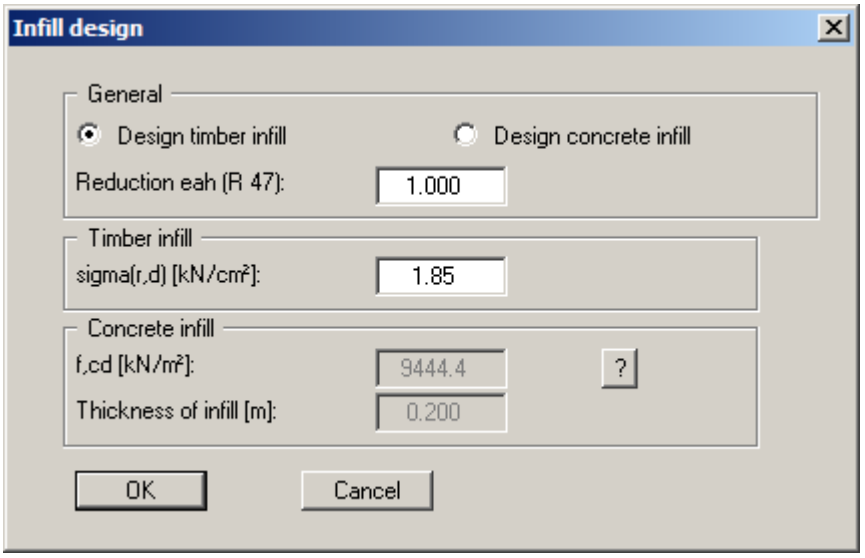


Enter the values in the dialog box and confirm with "**OK**". When the following message box opens, confirm.

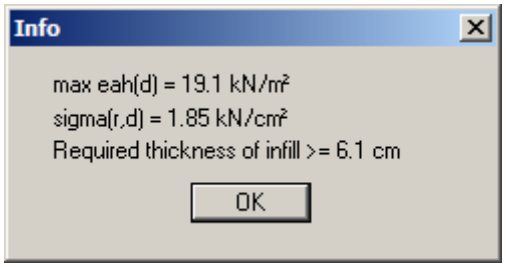


The design values for the soldier pile are then displayed in a message box. The data can be copied to a report via the Windows clipboard, for example, by pressing the "**To clipboard**" button.

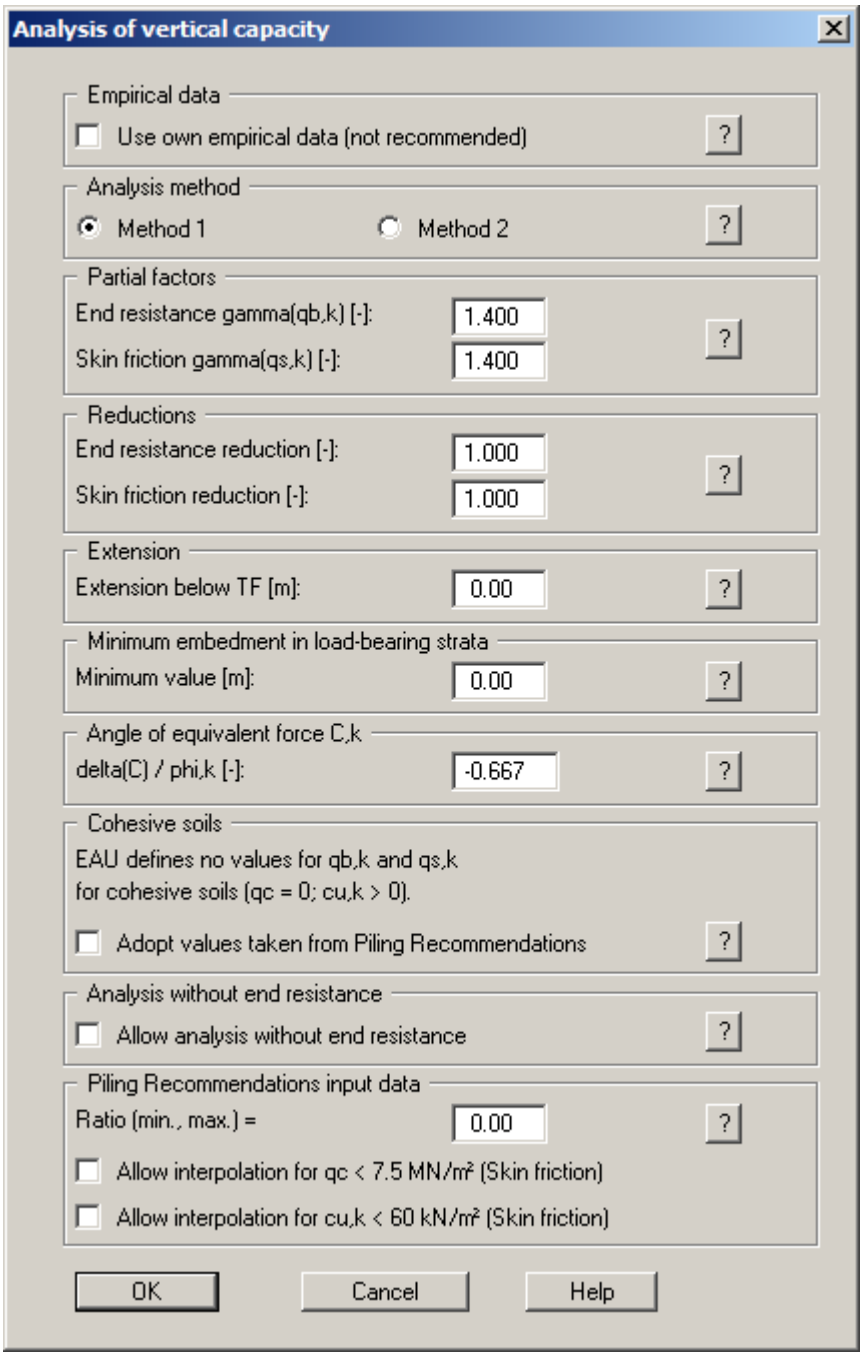
After leaving the result box, the infill walling design dialog box opens:



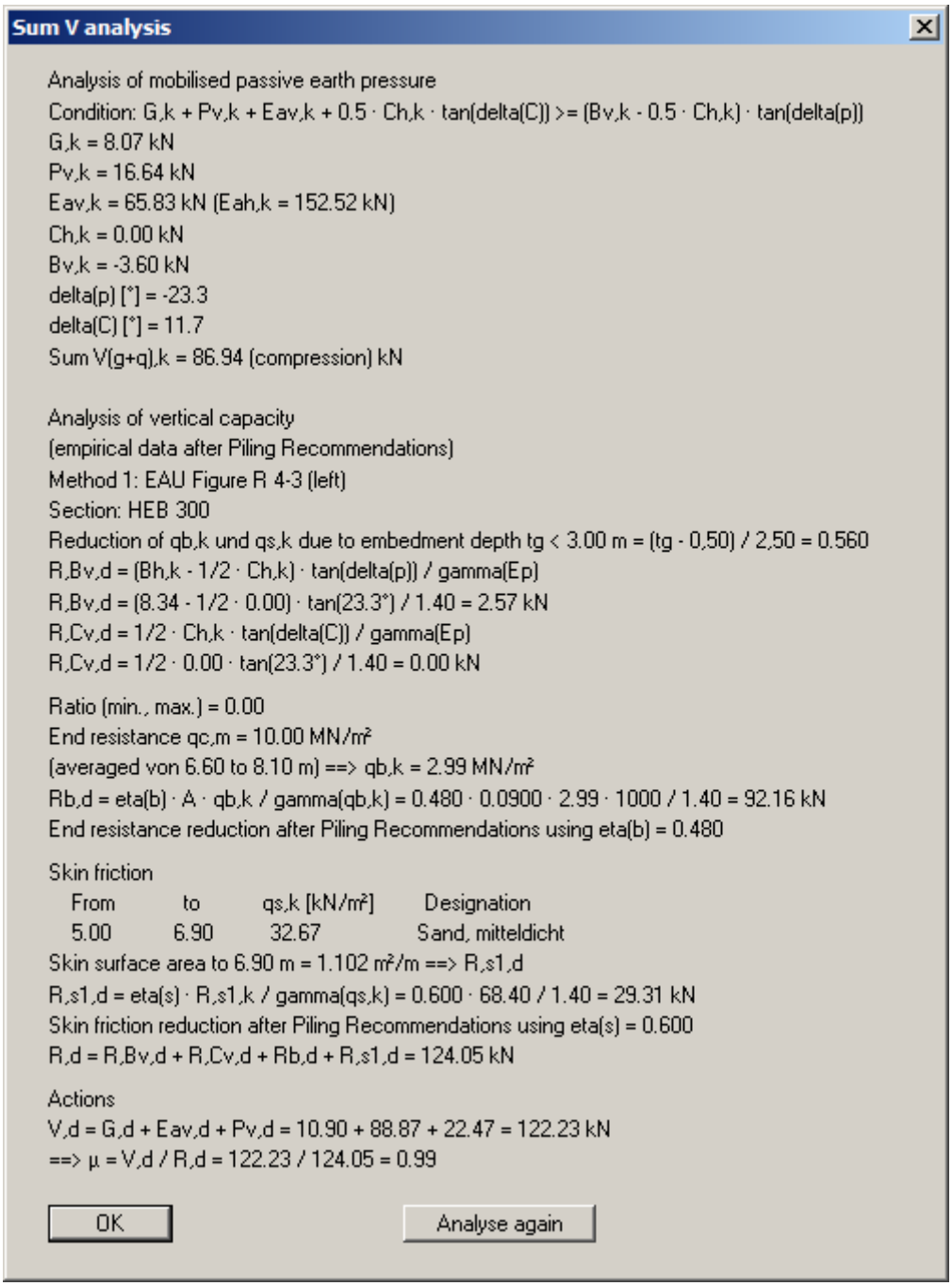
The result is displayed in a message box after pressing "**OK**":



The check boxes activated by default in menu item "**Editor 1/Verifications/Partial factors**" are "**Perform 'Vertical capacity' analysis (EAU, EAP and EAB)**" and "**Perform 'Sum V' analysis**". The following dialog box therefore opens once the infill walling has been designed:



A minimum embedment in load-bearing ground is not demanded in the EAU and EAB. The specified value of ***2.50 m*** is guided by the Recommendations on Piling (*EA-Pfähle*). You can therefore set the value for the minimum embedment depth to ***0.0 m***.



After clicking the "**Analyse again**" button it is possible to make changes to the above dialog box in order to analyse the vertical capacity and have the analysis performed again.

Leaving the box with "**OK**", the earth pressure distribution, the moment, shear force profile and normal force profiles, as well as the bending line, will be displayed on the screen. The analysis and design of the retaining wall are complete.

6.1.10 Step 9: Evaluate and visualise the results

Four legends appear on the screen, containing the soil properties, the main elements on which the analysis is based, and the main design results. The fourth legend contains a diagram of the retaining wall (see below).

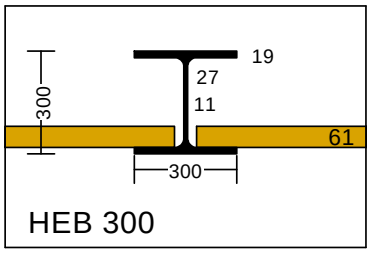
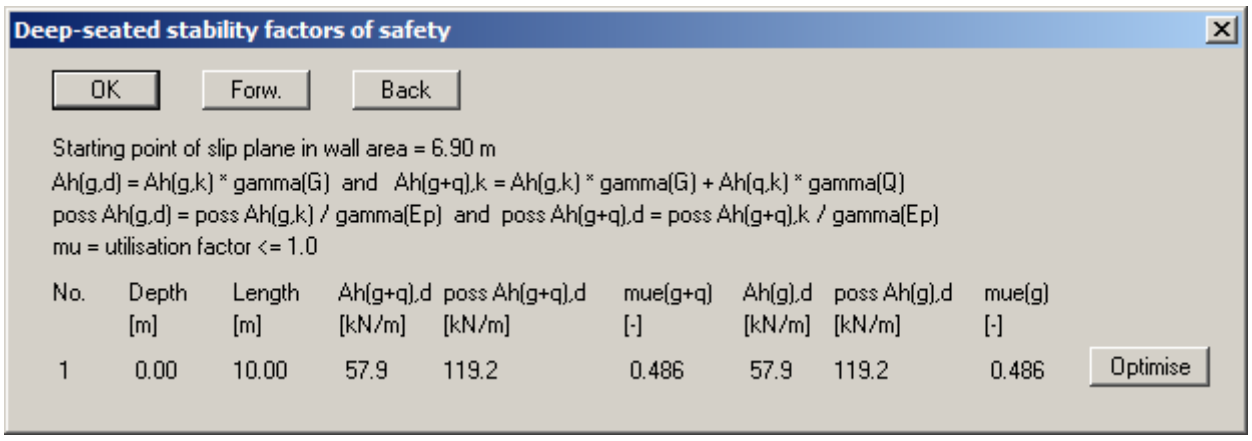


Figure 2 Retaining wall diagram

The graphics can be printed on the selected printer ("**File/Print and export**" menu item). You can also print off a detailed protocol ("**File/Print output table**" menu item). The zoom function (see "**Graphics preferences/Zoom info**" menu item) allows you to magnify selected areas of the graphic. Double-clicking in the graphics at a particular point will cause a box to appear containing the corresponding state variables.

Other forms of evaluation are possible from the "**Evaluation**" menu, which will display the selected type of earth pressure redistribution, the design parameters, the maximum values and the anchor forces. The menu item "**Evaluation/Deep-seated stability summary**" is of particular interest:



In this example an utilisation factor of 0.48 was determined. Clicking "**Optimise**" will adjust the anchor length accordingly. The anchor length necessary for an utilisation factor of 0.99 will be displayed after a few seconds. Changing anchor length radically alters the structural system, since the total axial stiffness of the anchor is increased. Thus, following optimisation, a corresponding warning box appears. In practice, the effect of the changes on cross-section through optimisation is usually small and can be neglected. However, if you are unsure, it is better to reanalyse.

If you wish to add explanatory text or graphic elements, you can do so using the "**Mini-CAD**" module. You can save your work in a file by clicking "**File/Save as**".

6.2.1 Initial system

Several years ago a 10.25 m long sheet pile wall (AU 23; S 240 GP) was installed in a harbour. At the top of the sheet pile wall a very steep, 0.30 m thick armour was subsequently constructed. The gradient is 1 : 1.30. The armour is stable if a horizontal load of 10.7 kN/m and a vertical load of 5.0 kN/m can be accepted at the top of the sheet pile wall.

A residential building was erected together with the armour at the rear of the sheet pile wall, built on a pile foundation in order to avoid additional loading on the sheet pile wall.

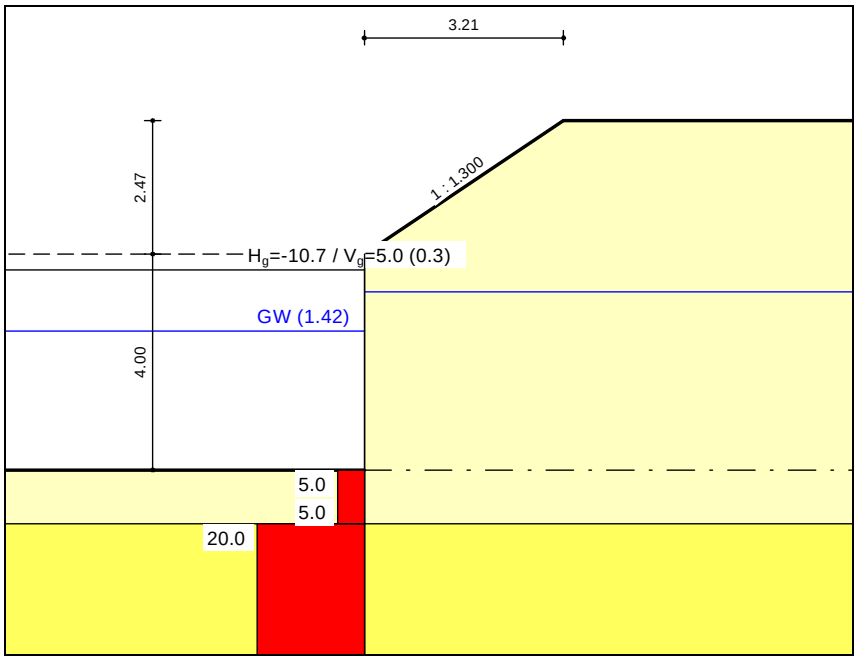


Figure 3 Illustration of worked example 2

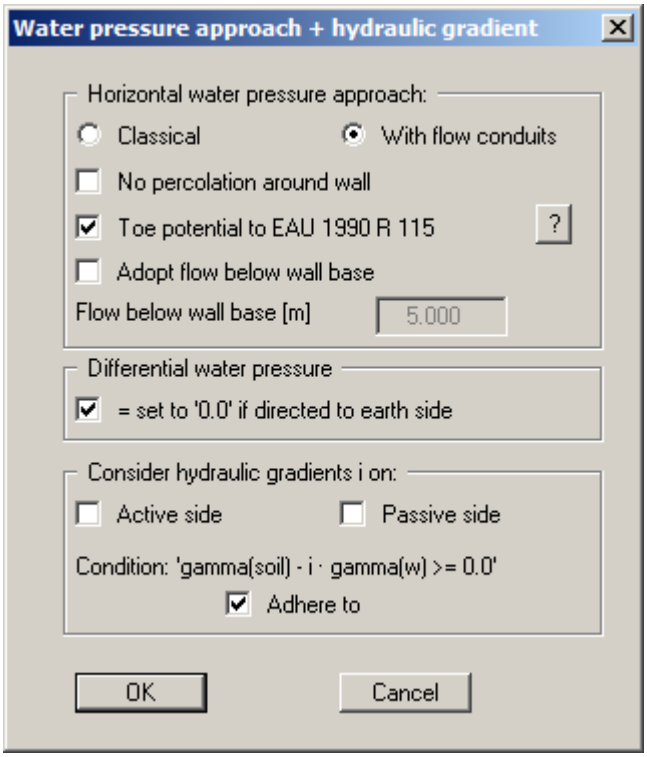
This example is included in the file "**Manual example 2-1.vrb**" in the program's *Examples* folder. Once you have opened the file you can follow the input described in the previous example in the relevant menu items in "**Editor 1**" and "**Editor 2**".

The sheet pile wall is analysed as elastically bedded (menu item "**Editor 2/Subgrade reaction moduli**").

Subgrade reaction moduli

Depth [m]	Top [MN/m³]	Bottom [MN/m³]	μ_{ks} [-]
0.00 - 1.00	5.000	5.000	0.1000
1.00 - 12.00	20.000	20.000	0.1000

To avoid unrealistically high differential water pressures the subsurface hydraulics are investigated using flow conduits (menu item "**Editor 1/Groundwater**").

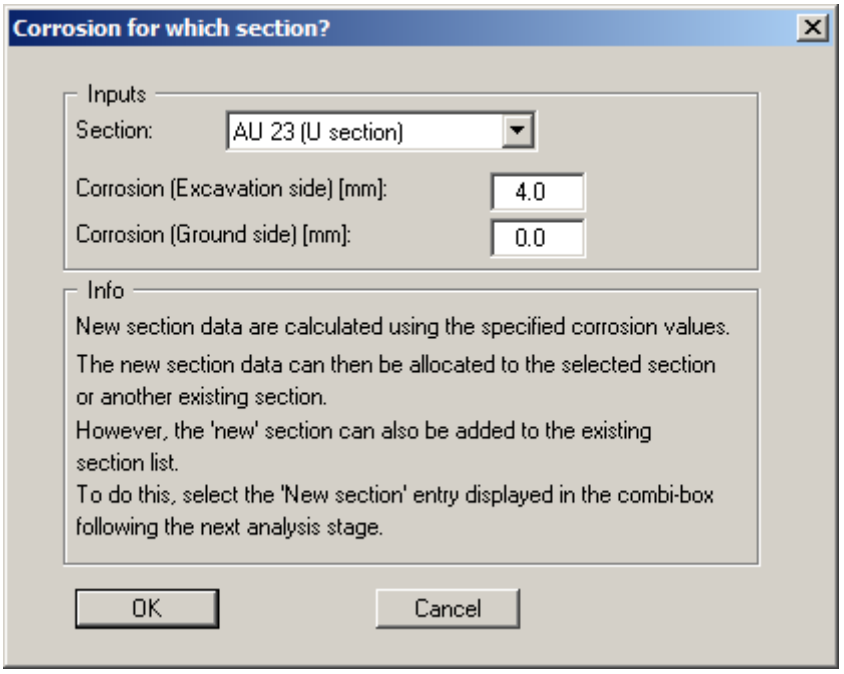


Under homogeneous conditions the check box "**Toe potential to EAU 1990 R 115**" must also be activated.

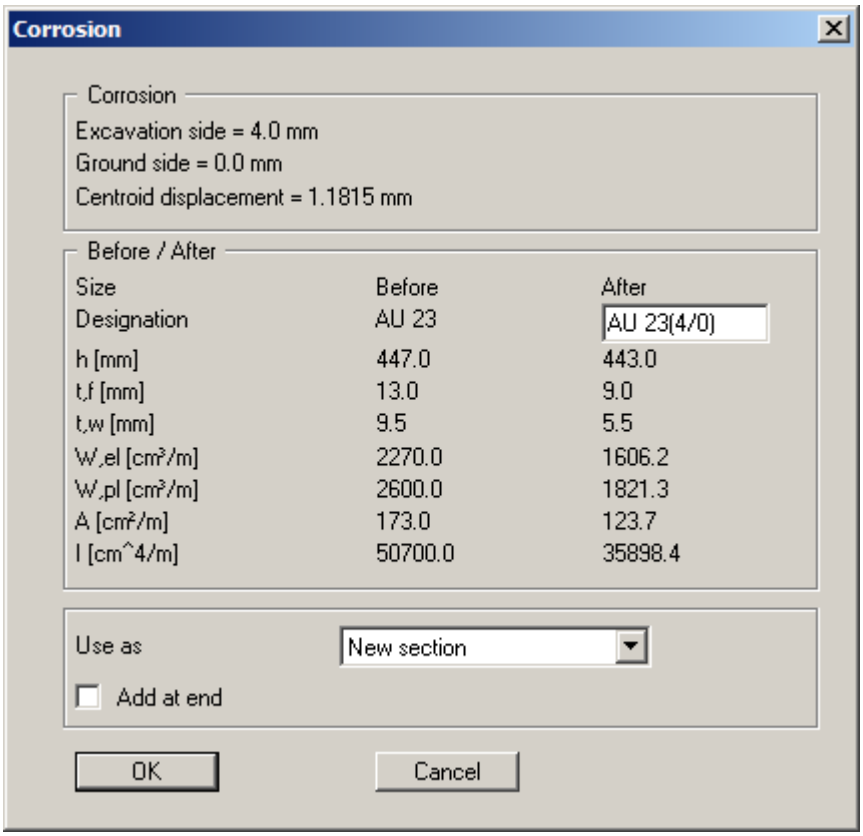
The new system can be verified using the specifications described above.

6.2.2 System with corrosion

However, an additional 4 mm of corrosion on the water side must be investigated. In "**Editor 2/Sections**" there is a "**Simulate corrosion**" button opening the following dialog box:.



After selecting the section and entering the corrosion, new section data are calculated and a new section name proposed.

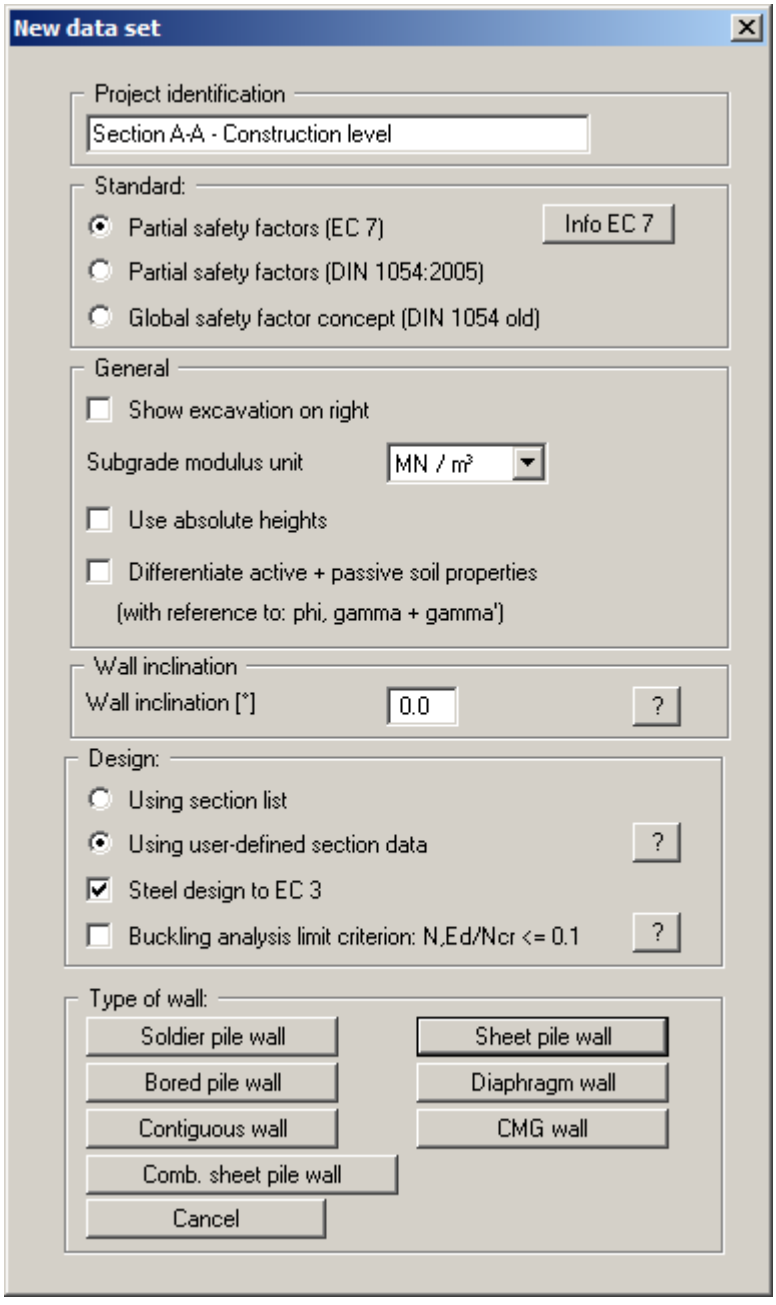


EAU 2012 8.1.8.4 stipulates that analysis may be based on DS-A. Analysis still fails using the newly calculated section data (see file "**Manual example 2-2.vrb**").

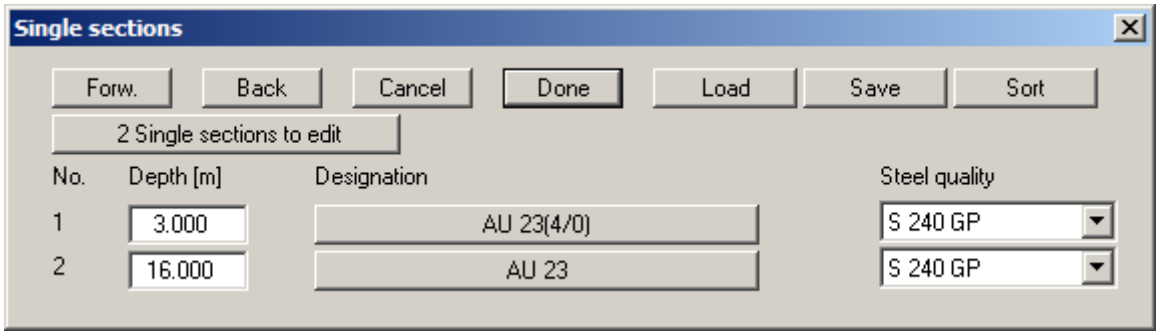
6.2.3 System with partial corrosion

The section is not corroded along its whole length, only in the low-water zone.

A second analysis using user-defined section data is performed.



Once the "**Using user-defined section data**" check box is activated the following system can be investigated ("**Editor 2/Section data**"):



4 mm of corrosion are assumed down to 3 m. Below this the section remains undamaged.

Using these specifications analysis is successful (file "**Manual example 2-3.vrb**").

7 Theoretical principles

7.1 General notes to types of retaining wall

The following types of retaining wall can be analysed:

- soldier pile walls;
- sheet pile walls;
- bored pile walls;
- diaphragm walls;
- contiguous pile walls;
- FMI wall;
- combined sheet pile walls.

Sheet pile walls, bored pile walls and diaphragm walls differ only in respect to the design of their sections, i.e. design of the reinforced concrete to EC 2 (bored pile wall = circular cross section; diaphragm wall = rectangular cross section). Soldier pile walls and contiguous pile walls differ only with respect to the design of the steel section (soldier pile wall) and the circular cross section (contiguous pile wall). The CMG (cut-mix-grout) wall, which is often also referred to as a MIP (mixed-in-place) wall, represents a special case.

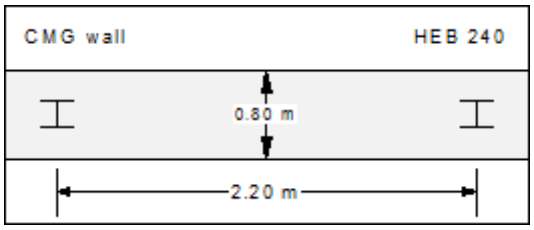


Figure 4 CMG (cut-mix-grout) wall

7.2 Soil properties

A maximum of 50 soil layers can be taken into consideration. The following parameters must be given for each:

- depth in metres below top of the wall, or absolute depth;
- unit weight [kN/m³] of moist soil γ ;
- unit weight [kN/m³] of buoyant soil γ' ;
- friction angle [°];
- cohesion (active and passive) [kN/m²];
- active angle of wall friction as the ratio δ_a/φ ;
- passive angle of wall friction δ_p/φ ;
- permeability to the left and right of the retaining wall [m/s];
- skin friction qs,k [kN/m²];
- cone resistance from CPT qc ,
- shear strength of the undrained soil cu,k .

If you activate the "**Differentiate active + passive soil properties**" check box in the dialog box in "**File/New**" or "**Editor 1/Analysis options**", you can enter differing friction angles and unit weights for the active and the passive sides.

Permeability is taken into consideration in a calculation of subsurface hydraulic conditions via flow conduits parallel to the wall (see Section 7.7.1.2).

If you want to analyse the pull-out resistance of anchors, activate the "**Verification with qs,k**" check box in the "**Editor 2/Anchors**" dialog box (see Section 8.3.7).

To analyse the vertical capacity to EAU, EAB and EAP, enter the cone resistance qc and the shear strength of the undrained soil cu,k.

7.3 Active earth pressure

Active earth pressure is analysed to DIN 4085. DIN 4085 provides two relationships for the coefficients of earth pressure k_{ah} (friction) and k_{ch} (cohesion). Alternatively, there is the option of determining the cohesion coefficient from $k_{ch} = k_{ah}^{-2}$, a method often found in older literature. **GGU-RETAIN** also provides the option of applying user-defined earth pressure coefficients, which can be entered in tabulated form for each type of soil.

7.4 At-rest earth pressure

The at-rest earth pressure coefficient is calculated compliant with DIN 4085:2011.

7.5 Increased active earth pressure

The coefficient of increased active earth pressure, k_{eh}, is obtained from the coefficient of active earth pressure and at-rest earth pressure:

$$\mathbf{k_{eh} = (1.0 - f) \cdot k_{ah} + f \cdot k_0}$$

$$0.0 \leq f \leq 1.0$$

7.6 Passive earth pressure

The coefficient of passive earth pressure can be analysed using a number of methods:

- DIN 4085:2011;
- DIN 4085:1987;
- Streck,
- Caquot/Kerisel;
- DIN 4085:1987/Caquot/Kerisel.
- Mohr/Coulomb (root)
  This approach is obsolete and should no longer be used!

Besides calculated values, you can also define your own.

The passive earth pressure acting on soldier piles is calculated after Weißenbach (EAB R 14). When the soldier piles are so close together that the effects of passive earth pressure overlap, the calculated values should be reduced. The passive earth pressure with and without overlapping must be determined for this purpose. Passive earth pressure acting on soldier piles without an overlap effect takes the shape of a parabolic curve and depends, amongst other things, on pile width. **GGU-RETAIN** calculates both values (with and without overlap effect) for all subdivisions. The lower of the two values obtained is then used in the subsequent analysis. Thus, in a graphic visualisation of passive earth pressure acting on a soldier pile, part of the curve may be parabolic, while another part is linear.

7.7.1 Approach for sheet pile walls and in-situ concrete walls

7.7.1.1 Classical water pressure approach

For sheet pile walls and in-situ concrete walls, water pressure can affect the active and passive sides of the wall. The classical water pressure approach is illustrated in the following figure:

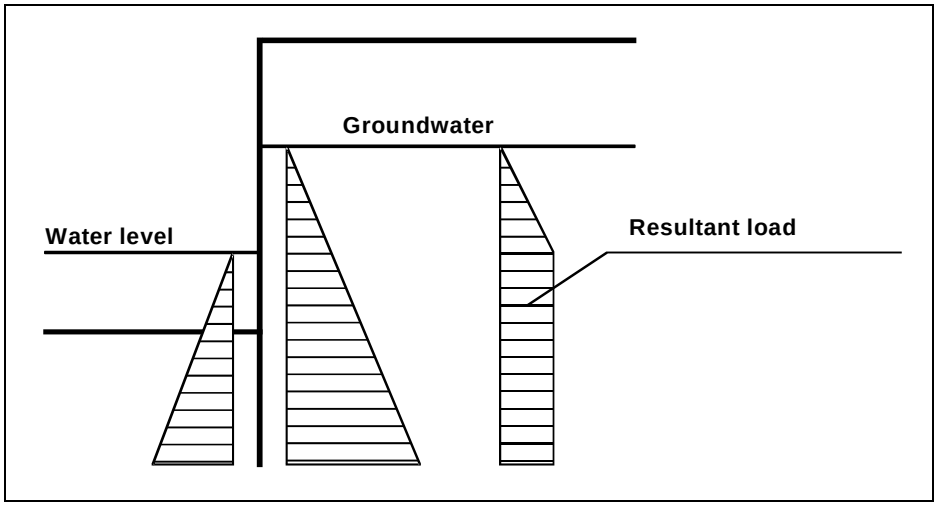


Figure 5 Classical water pressure approach

However, the classical approach does not take the permeability of the soil into consideration. The gradients on the active and passive sides are obtained by assuming a linear reduction in pressure around the retaining wall.

7.7.1.2 Water pressure approach using flow conduits

Beside the classical water pressure approach, **GGU-RETAIN** also allows analysis of walls around which groundwater can percolate. To facilitate this, a flow conduit is calculated on the active and on the passive side. The permeabilities within the flow conduit can be specified separately for each soil layer.

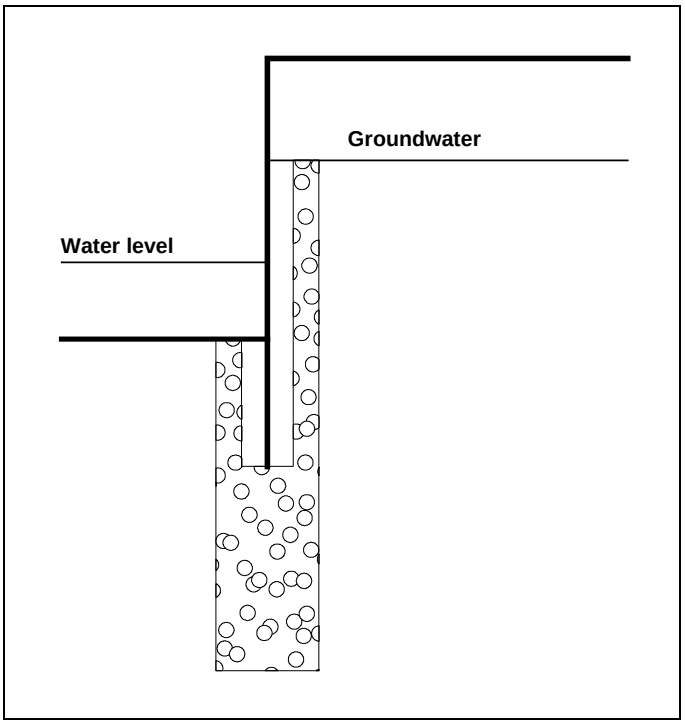


Figure 6 Water pressure approach using flow conduits

If you select the water pressure approach using flow conduits, the potentials *h* at the height of both water levels will be automatically adopted as boundary conditions. In the simplest case of a global permeability value the result is linear pressure dissipation along the flow conduit. Besides the water pressures, the hydraulic gradients will also be calculated and can be selected for consideration in the calculations of active and passive earth pressures.

An approach using flow conduits is, however, much more interesting for use with variable permeabilities. In this case, the water pressures are correctly determined in accordance with potential theory by employing a small finite-element module within **GGU-RETAIN**. You can still define additional potentials at any location to the left or right of the retaining wall. This allows correct, simple consideration of several groundwater storeys or confined aquifers. In addition, employing user-defined potential definitions allows you to create or model any kind of water pressure distribution.

7.7.1.3 Recommendation for water pressure approach

The classical water pressure approach is valid when the toe of the retaining wall is embedded in a layer of ***low permeability***. Otherwise the difference in water pressure at the wall toe (see Figure 5 Classical water pressure approach) has no physical meaning, but nevertheless provides conservative design values.

The classical water pressure approach and the approach using flow conduits (under uniform conditions of permeability) both assume a linear pressure drop along the wall. Given ***uniform permeability***, this can lead to an underestimate of the hydraulic gradient, since the two-dimensional flow to which the wall is subjected is not taken into consideration (see also EAU). Using flow conduits, with additionally defined potentials at the toe of the wall, this can be corrected.

For soils with ***non-uniform permeability*** above the wall toe, the water pressure approach using flow conduits should always be preferred. If doubts remain, you will have to carry out a two-dimensional groundwater analysis according to the EAU. The potentials determined at the wall can be incorporated into flow conduits.

If an ***impermeable*** layer occurs at the base of the wall there will be no hydraulic gradient along it. Nevertheless, when using the classical approach, a linear reduction in water pressure is assumed in the literature, which really amounts to unnecessarily doing the same thing twice.

7.7.1.4 Possible conceptual error using flow conduits

On the other hand, the following conceptual error can easily be made using flow conduits when dealing with the following system:

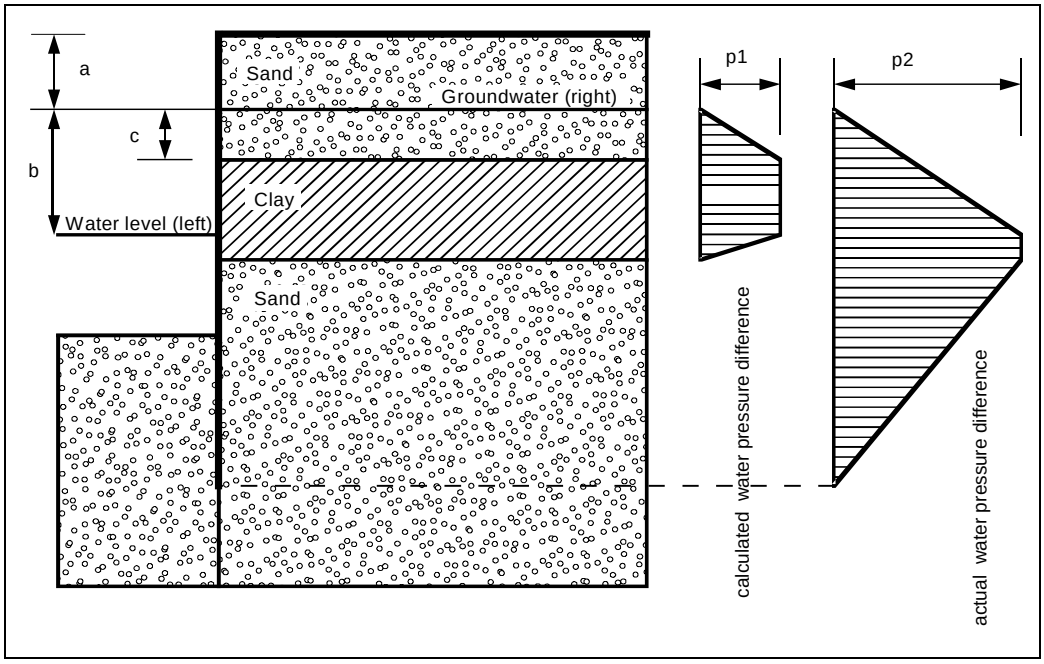


Figure 7 Possible conceptual error using flow conduits

$$\mathbf{p1 = \gamma_w \cdot c}$$

$$\mathbf{p2 = \gamma_w \cdot b}$$

The permeabilities of the clay and sand layers are $1 \cdot 10^{-8}$ m/s and $1 \cdot 10^{-4}$ m/s, respectively. Although **GGU-RETAIN** conforms completely to potential theory, the calculated difference in water pressure, p1 [kN/m²], does not correspond to the distribution of pressure that would be expected. Based on what has already been described, the application assumes a current of water in a flow conduit parallel to the wall from the water level on the right to the water level on the left of the wall. At the top right-hand end of the flow conduit the water level corresponds to the right-hand water level, while at the top left-hand end of the flow conduit it corresponds to the left-hand water level. The reduction in water pressure between these two points is now determined according to potential theory. The resulting difference in water pressure between the left and right sides corresponds to the distribution p1 shown in Figure 7, and according to the applied method is perfectly correct. However, if the groundwater level below the layer of undisturbed clay is the same as that above it, this approach will produce an erroneous value for the water pressure distribution. In the above example, the water pressure below the layer of clay will be determined almost entirely by the water pressure in the excavation. If the water pressure below the layer of clay is to correspond with that above it, using flow conduits, a potential of

$$\mathbf{h = a}$$

must be defined on the right side, below the layer, so as to achieve the desired distribution of water pressure differentials, p2, as shown in Figure 7. Although **GGU-RETAIN** conducts plausibility checks, there are nevertheless situations in which an analysis without additional definition of a potential below the clay layer is useful. Whether or not the computed difference in water pressure is what you had in mind can be easily checked by selecting the "**Differential water pressure**" command button in the dialog box that appears when you click "**Graphics output preferences**" in the "**System**" menu. Then **GGU-RETAIN** graphically displays the difference in water pressure between the left and right sides of the retaining wall.

7.7.2 Approach for soldier pile walls

In general, adopting water pressures for soldier pile walls does not make sense, because they are usually permeable to water. But even when using timber infill, water pressure can build up behind the wall in certain circumstances (see Figure 8).

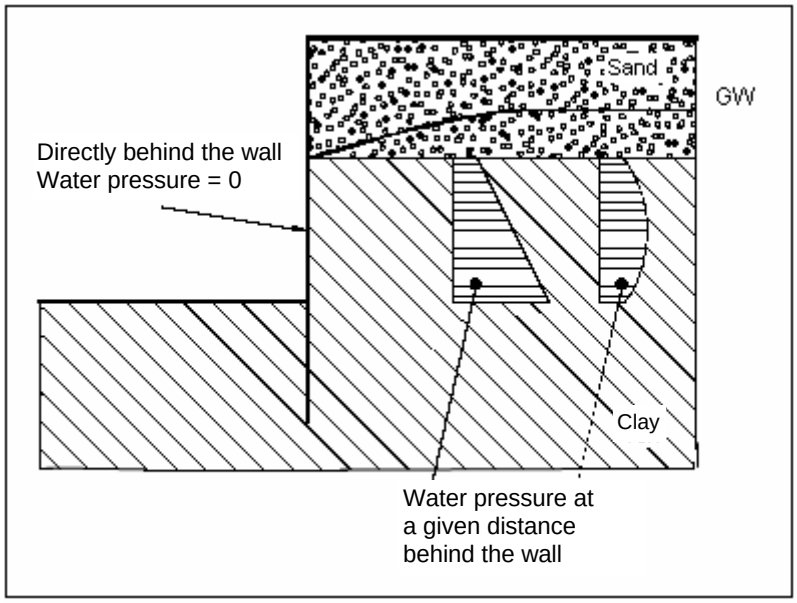


Figure 8 Water pressure for soldier pile walls

The program therefore allows for adopting water pressures with soldier pile walls.

7.7.3 What is a potential ?

In subsurface hydraulics the potential (h) is composed of:

- the elevation headdefinition (y) of the point under consideration (in metres) and
- the water pressure (u) at the point under consideration (in metres).

If p is the water pressure in kN/m² at a particular point, then water pressure, u (in metres) = p/γ_w (γ_w = unit weight of water ≅ 10.0 kN/m³).

$$\mathbf{h = p/\gamma_w + y}$$

h = potential [m]
p = water pressure [kN/m²]
y = elevation head [m]

An example:

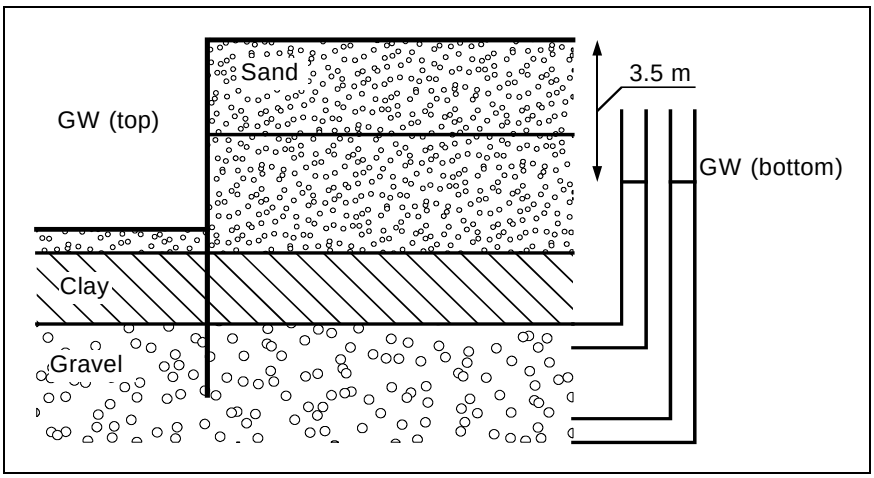


Figure 9 Definition of potential

In this example, two permeable soil layers are separated by a low-permeability layer (clay). In the excavation, groundwater is kept at the base level. The groundwater in the gravel layer below the clay is under pressure and rises to 3.5 m below the top of the wall (represented in the diagram by the water levels in the two standpipes on the right). The lowest point in the right-hand standpipe is at a higher water pressure than that in the left-hand standpipe, but both have the same potential of 3.5 m below the top of the wall. Because potential is a function of the elevation head and the water pressure [m], the left-hand standpipe makes up for its lower water pressure by its greater elevation head.

If you want to calculate such a system using flow conduits, it is sufficient to enter a potential of 3.5 m below the top of the wall for any point within the layer of gravel. The potential above the layer of clay is defined by groundwater levels to the left and right of the wall. **GGU-RETAIN** automatically applies these conditions, which it uses, together with the permeabilities, to calculate the potentials along the flow conduit. From the calculated potentials (h), water pressure (p) and also the gradient (i) can now be determined. Applying the above formula for p:

$$\mathbf{p = \gamma_w \cdot (h - y)}$$

For the gradient (i) we have:

$$\mathbf{i = \Delta h / \Delta L}$$

where Δh = potential difference between two points
ΔL = distance between the two points

Using flow conduits, the load-decreasing effect of the upwardly directed hydraulic gradient on passive earth pressure can now be taken precisely into consideration.

GGU-RETAIN can handle up to 20 berms on both the active and the passive sides of the wall. The berms can include surcharges. The effect on earth pressure is taken into consideration according to the Piling Handbook (Krupp Hoesch Stahl).

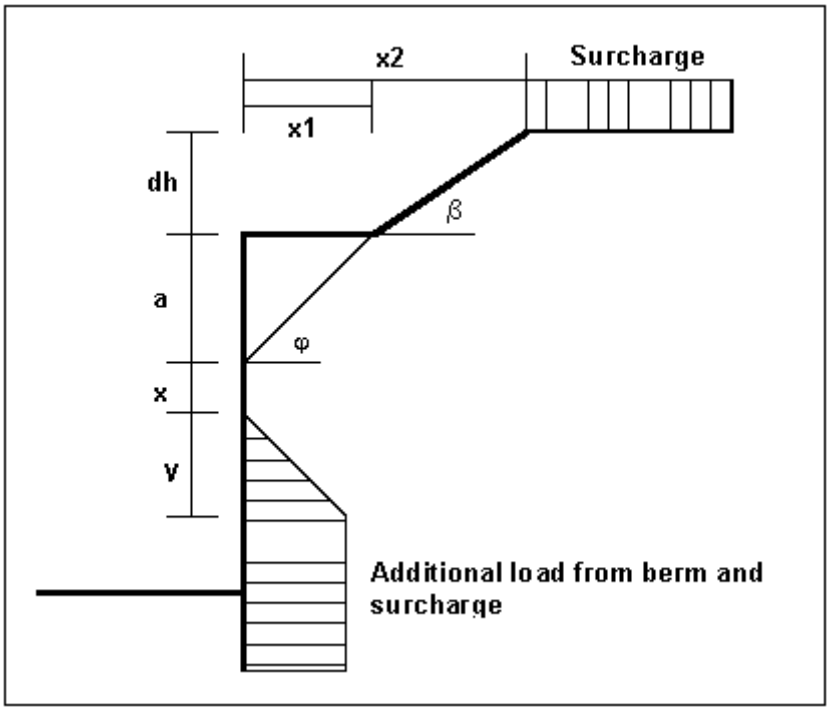


Figure 10 Berms on the active side

The following relationships apply for the parameters x and y:

$$\mathbf{x = k_{ah0} / (k_{ah\beta} - k_{ah0}) \cdot a}$$

$$\mathbf{y = k_{ah0} / (k_{ah\beta} - k_{ah0}) \cdot x}$$

$$\mathbf{\Delta e_{ahu} = \gamma \cdot dh + surcharge}$$

γ = unit weight of soil in the berm area

If the angle β is greater than φ, it is assumed that $\beta = \varphi$ for analysis. Berms on the passive side are dealt with in exactly the same manner.

7.9 Area loads

Up to 20 area loads can be positioned on the active side at any height.

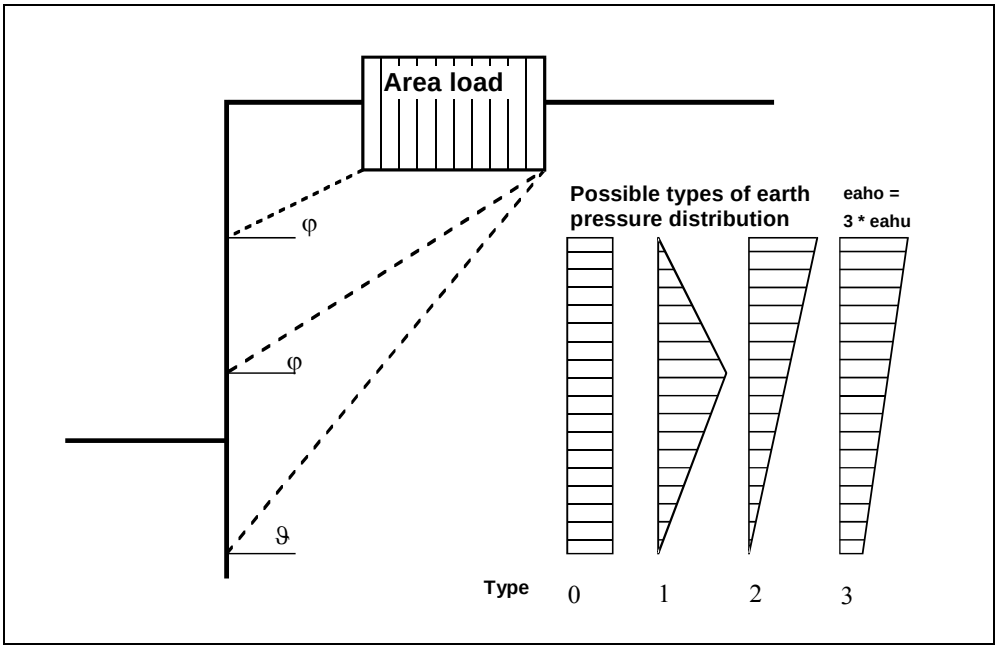


Figure 11 Area load

The slip surface angle for the active earth pressure resulting from the self-weight of the soil is adopted for analysis compliant to DIN 4085.

$$\vartheta_{ag} = \varphi + \arctan\left(\frac{\cos(\varphi - \alpha)}{\sin(\varphi - \alpha) + \sqrt{\frac{\sin(\varphi + \delta_a) \cdot \cos(\alpha - \beta)}{\sin(\varphi - \beta) \cdot \cos(\alpha + \delta_a)}}} \right)$$

When there are a number of soil layers, **GGU-RETAIN** moves from layer to layer applying the appropriate angles of friction. The type of resulting earth pressure distribution can be defined in 4 different ways.

For at-rest earth pressure, the area loads are calculated by increasing by the factor k0/kah in compliance with DIN 4085:2011 Section 6.4.3.

Alternatively, it is possible to determine the demands placed on a wall from area loads using the theory of elastic half-space. The two load concentration factors '**3**' and '**4**' can be taken into consideration (see also Figure 12).

- For overconsolidated, cohesive soils the concentration factor "**3**" applies, where:
  $e_{op} = q/\pi \cdot (\beta_2 - \beta_1 + \cos\beta_1 \sin\beta_2 - \cos\beta_2 \sin\beta_2)$
- For non-cohesive soils, or for cohesive but not over consolidated soils, the concentration factor "**4**" applies, where:
  $e_{op} = q/4 \cdot (\sin^3\beta_2 - \sin^3\beta_1)$

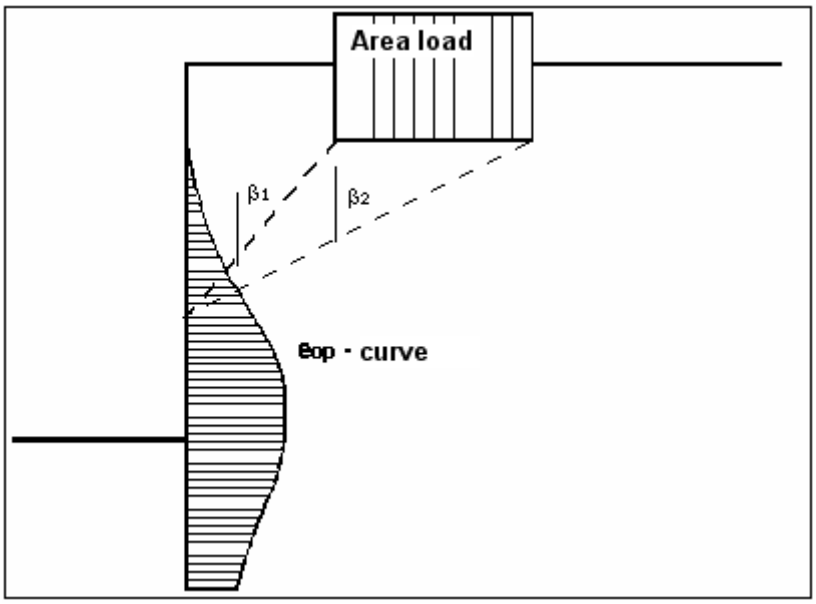


Figure 12 At-rest earth pressure from area loads

With regard to the kind of earth pressure, area loads can be defined independent of the global preferences (see menu item "**Editor 1/Type of earth pressure**", Section 8.2.6).

It is possible to enter a horizontal load p(h) when defining area loads ("**Editor 2**" menu, Section 8.3.2).

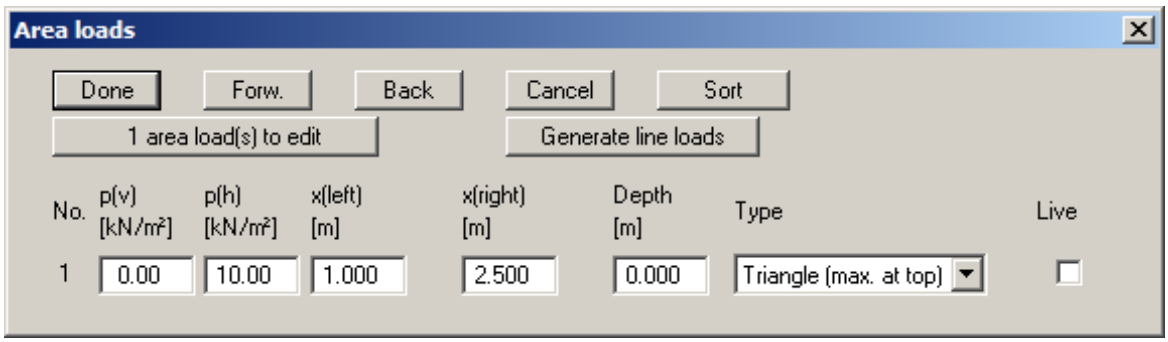


Figure 13 Horizontal loads p(h) in area loads

In this case, a horizontal load of 10 kN/m² was defined with a width of 1.50 m. This corresponds to a horizontal load H of 15 kN/m. According to the Piling Handbook (*Spundwand-Handbuch*) this results in the active earth pressure shown in Figure 14 for homogeneous ground:

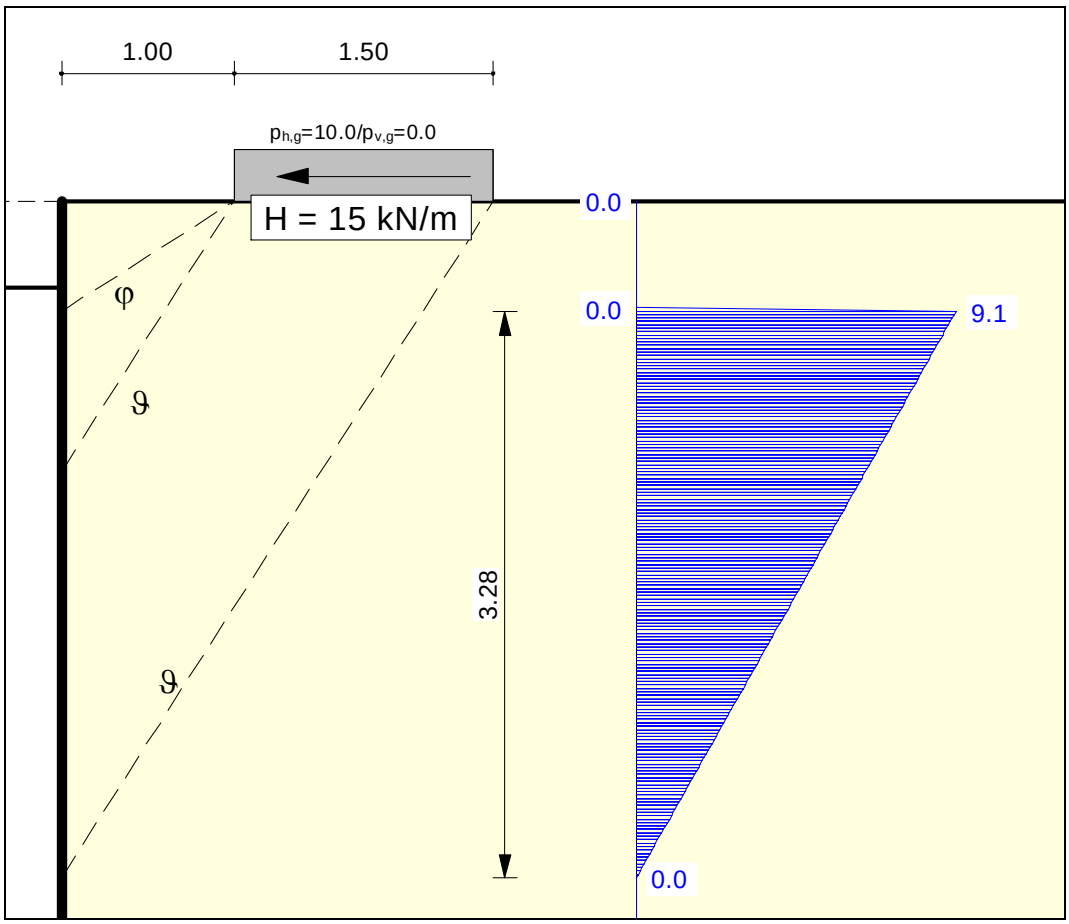


Figure 14 Active earth pressure from horizontal loading in homogeneous ground (φ = 32,5°)

The area covered by the active earth pressure resulting from horizontal loads corresponds to a horizontal load of 15 kN/m.

In homogeneous ground, the resulting earth pressure is independent of unit weight and cohesion. The earth pressure distribution with depth is acquired from the friction angle φ and from ϑ, which results from it. The earth pressure area is independent of the angle of friction and always corresponds to the adopted horizontal force H. In stratified ground the beginning and end of the earth pressure's zone of influence is acquired by averaging φ and ϑ.

If at-rest earth pressure or increased active earth pressure are adopted the horizontal forces from the resulting earth pressure cannot be greater than the acting horizontal force H for equilibrium reasons. The same earth pressure distribution is therefore adopted in **GGU-RETAIN** for at-rest earth pressure and increased active earth pressure as for active earth pressure.

7.10 Line loads

Line loads perpendicular to the retaining wall axis (for example, from loads exerting a vertical force on the wall) are treated as shown in Fig. 4.20 on page 64 of the Piling Handbook (*Spundwand-Handbuch 1977*). Data is entered in the form of a number of discrete area loads.

7.11 Bounded surcharges (active side)

Up to 20 bounded surcharges can be positioned at any height on the active side.

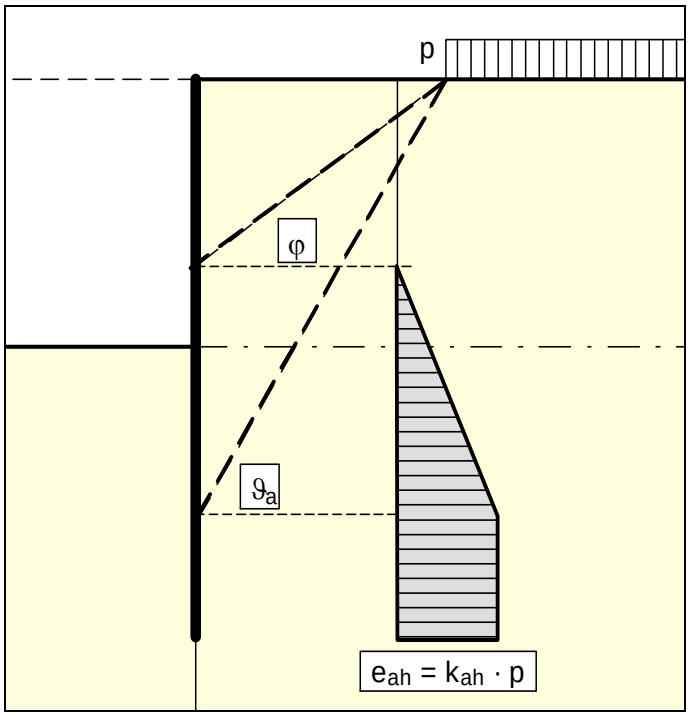


Figure 15 Bounded surcharge (active side)

The earth pressure coefficient k is acquired from k_{ah} for active earth pressure and from k_0 for at-rest earth pressure. If this option is activated, the resulting earth pressure is then redistributed.

7.12 Double-bounded surcharges (active side)

If two bounded surcharges are defined as follows:

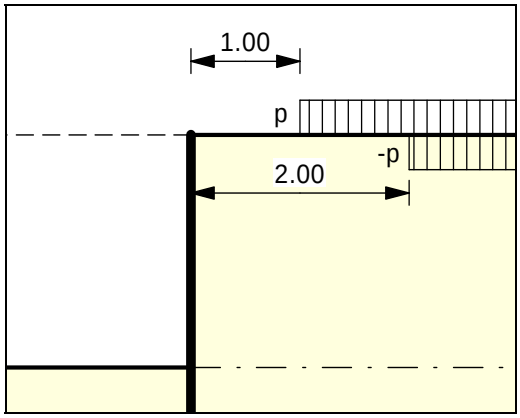


Figure 16 Two bounded surcharges

- Bounded surcharge 1:
  begins at x = 1,00 m and has the value p.
- Bounded surcharge 2:
  begins at x = 2,00 m and has the same value as surcharge 1 but with a negative sign.

A double-bounded surcharge, which acts as a consequence of p at x = 1.00 to x = 2.00, is thus defined as the result of earth pressure analysis. The analysis result is correct in terms of the adopted approach. However, the graphics are not really satisfactory and may be irritating for an examiner. Double-bounded surcharges were therefore introduced.

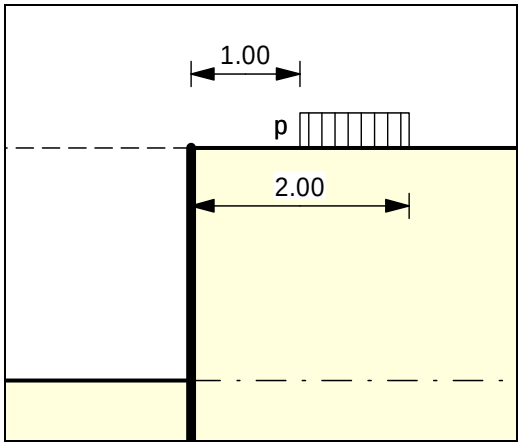


Figure 17 Double-bounded surcharge

The same result is achieved as for the definition provided by Figure 16 but with improved graphics.

The area loads described in Section 7.9 are based on different assumptions for the resulting earth pressure, so the definition of a double-bounded surcharge does not provide the same result as an equivalent area load. Generally speaking, the earth pressure from double-bounded surcharges is less than that from area loads.

7.13 Bounded surcharges (passive side)

Up to 20 bounded surcharges may be adopted at any height on the passive side. The passive earth pressure is computed as follows:

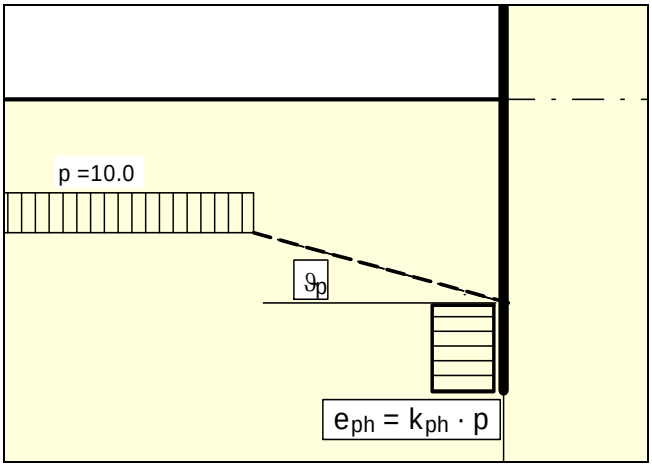


Figure 18 Bounded surcharge (passive side)

7.14 Double-bounded surcharges (passive side)

Double-bounded surcharges on the passive side can be modelled using two bounded surcharges (also see Section 7.12). Double-bounded surcharges on the passive side were only implemented due to the better graphical representation.

7.15 Length surcharge

In the case of full or partial fixity of the retaining walls, a length surcharge for the theoretically determined embedment depth is required. In accordance with the EAB, this surcharge can be adopted at 20% of the theoretically determined embedment depth for full fixity. For partial fixity the value is acquired by linear interpolation between 0% and 20%. Alternatively, the length multiplier Δx can also be calculated in accordance with the EAU (*Empfehlungen des Arbeitsausschusses "Ufereinfassungen"* - Recommendations of the Committee for Waterfront Structures, Harbours and Waterways):

$$\Delta x \geq C_h / e_{phC}$$

C_h = equivalent force after Blum (horizontal component)
e_{phC} = passive earth pressure stress on the equivalent force side below the theoretical toe TF

GGU-RETAIN can use either method. When calculating with the above equation, **GGU-RETAIN** always adopts the coefficient of passive earth pressure after Streck. After determining Δx, the program checks whether the additional length extends into soil with a lower coefficient of passive earth pressure, in which case it uses the lower value. The output table will provide you with a detailed presentation of the results.

7.16 Structural system

The structural analysis required to determine the state variables (displacement, moment, shear force and normal force) is not carried out by means of a continuous beam analysis, as is usual in retaining wall applications, but by means of a rod construction module, which treats the retaining wall and any anchors and struts as a single structural system. For inclined anchors in particular, the interactions between the anchor and the retaining wall can thus be correctly processed in a single computation. In a system with one anchor and one strut, for example, the following schematic would represent the governing structural system:

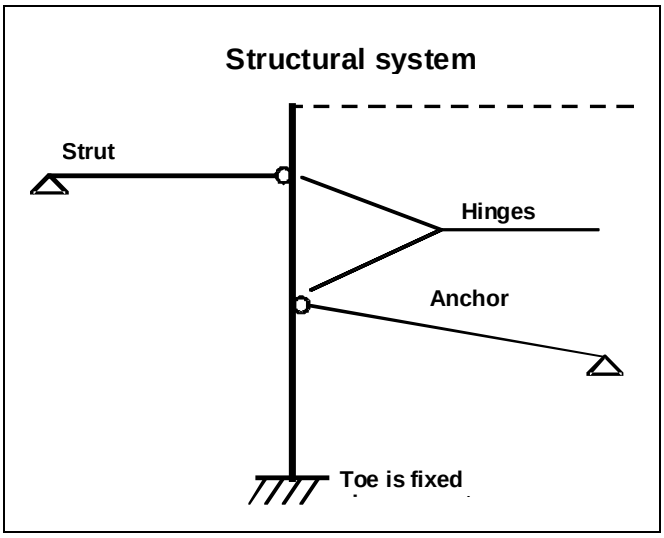


Figure 19 Possible structural system

In addition, a vertical load can be defined for a strut and, if necessary, a rigid connection to the retaining wall be taken into consideration. For anchors and struts, the axial rigidity must be specified so that its influence on the action effects can be correctly recorded. The rod construction module can also compute using 2nd order theory. This means that buckling length investigations for struts, for example, can be dispensed with.

The theoretical principles of the rod construction module can be traced back to an article by Duddeck/Ahrens (*Betonkalender* 1976 (Concrete Calendar), Volume 2). Basically, this is a finite element method based on the displacement method. The division of the retaining wall and the struts into rod sections (finite elements) can be controlled by input. However, the precision of the analysis does not depend on the size of the divisions. If the subdivisions are smaller, you will simply get more intermediate values for the state variables. The rod divisions can only exert an influence on the result of the subgrade reaction or on an analysis using 2nd order theory. However, confirmation prompts are programmed in to prevent impermissible values being entered.

Using FEM produces equation systems in which the number of unknown quantities is a function of the number of rods. The solution to the equation system is acquired in **GGU-RETAIN** using Cholesky's method, which is also employed in other GGU programs and is highly stable numerically. Nor have there been any numerical difficulties reported from other GGU applications.

7.17 Design

When you start an analysis, the program requires the stiffnesses of the retaining wall and any anchors and struts present. This data can be specified before going ahead with the analysis. The **GGU-RETAIN** program also provides an additional option for determining the optimum section for the computed action effects, using a list of sections loaded with the program. This list can also be edited and expanded. To do this, the program also requires you to enter the maximum allowable stress. If the newly determined section does not coincide with the section on which the analysis was based, **GGU-RETAIN** alters the calculated displacements as a function of the moments of inertia of the two sections. However, a new analysis would normally be performed, as the stiffness conditions have changed, because the axial rigidity of the anchors and struts, and any subgrade reaction moduli, do not alter. In most cases, nevertheless, renewed analysis is actually unnecessary, because the axial rigidity of the anchors/struts is generally so great that appreciable support displacements do not occur. In the appropriate program situations, you will still be warned.

Stress determination can be by one of three different means:

- working $\sigma = N/A + (N \cdot w + M)/W$;
- working $\sigma = N/A + M/W$;
- working $\sigma = M/W$.

where

N = greatest normal force (absolute)
A = cross-sectional area
w = greatest displacement (absolute)
M = greatest moment (absolute)
W = section modulus.

Shear design is also performed.

In-situ concrete walls can be designed to EC 2. Shear design is also possible here.

7.18 2nd order theory

The differential equation for a ***normal*** flexural member is:

$$EI\, w''''(x) = q(x)$$

The normal force N is taken into consideration for a buckling member:

$$EI\, w''''(x) + N\, w''(x) = q(x)$$

Analysis is performed on the deformed system. In

- **DIN EN 1993-5**
  Design of Steel Structures,
  Part 5: Piling

analysis using 2nd order theory is recommended for analysis of sheet pile walls under buckling loads and

- **DIN EN 1993-1-1**
  Design of Steel Structures,
  Part 1-1: General Rules and Rules for Buildings

referred to. Analysis using 2nd order theory produces more accurate results than the usual and simplified equivalent member method. Analysis of sheet pile walls under buckling loads is performed in **GGU-RETAIN** compliant with DIN EN 1993-1-1.

Analysis using 2nd order theory requires a predeformation or pre-curvature of the underlying system. Pre-curvature values are given in Table 5.1 of DIN EN 1993-1-1.

Table 5.1 – Pre-curvature design values $e_{0,d}/L$ for structural elements

Buckling line from Table 6.1	Elastic analysis	Plastic analysis
	$e_{0,d}/L$	$e_{0,d}/L$
a_0	1/350	1/300
a	1/300	1/250
b	1/250	1/200
c	1/200	1/150
d	1/150	1/100

Pre-curvatures are given as a function of the buckling line. In simplification, sheet pile walls can be analysed with a pre-curvature of $e_{0,d}/L = 1/150$.

In embedded, non-anchored walls the deformed system is defined by an inclination of the wall.

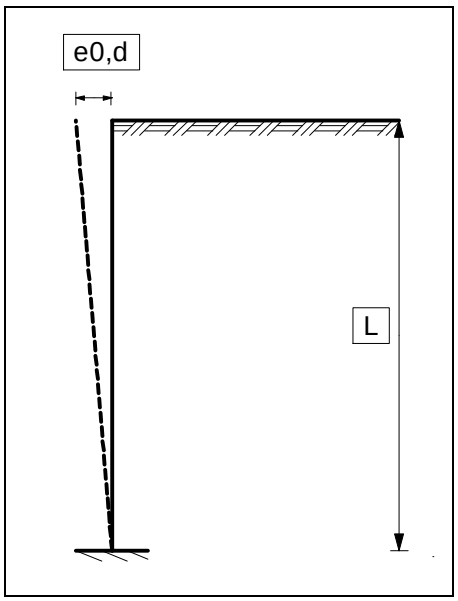


Figure 20 Embedded, non-anchored wall

In embedded, singly-anchored walls or walls with a free earth support, the deformed system is defined by a linear pre-curvature from the support point to the top of the wall and a parabolic pre-curvature between the support points and the foot of the wall.

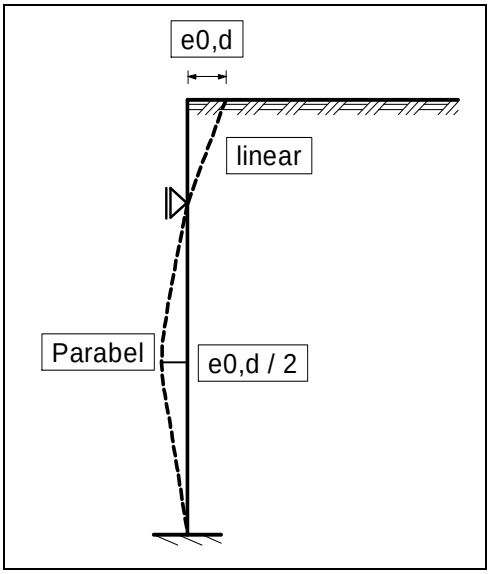


Figure 21 Singly-anchored, embedded or wall with free earth support

The length L is given per field. The procedure for two rows of anchors is shown in Figure 22.

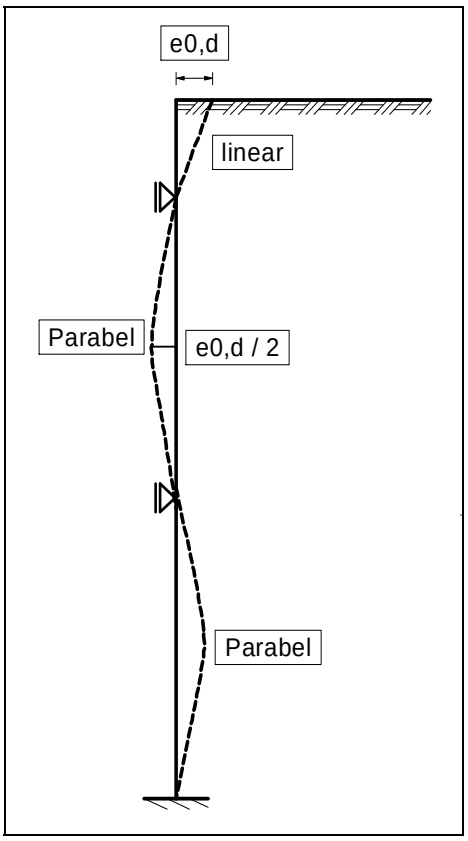
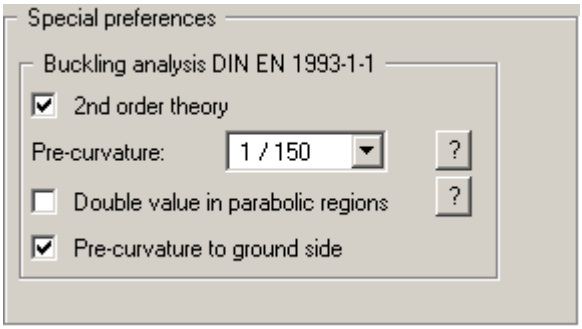


Figure 22 Doubly-anchored, embedded or wall with free earth support

The settings are defined in **GGU-RETAIN** in the analysis menu under "**System/Analyse**".



The size and direction of the pre-curvature can be specified. Whether a pre-curvature towards the ground side or the atmosphere side provides the more unfavourable design values is system-dependent: the following message will therefore be displayed once analysis begins:

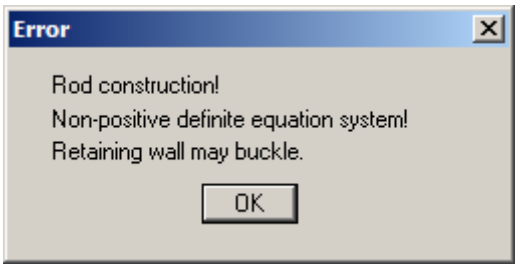
You must therefore deactivate the "**Pre-curvature to ground side**" check box following successful analysis and check, in a new analysis, whether pre-curvature to the atmosphere side delivers less favourable values.

In an analysis using 2nd order theory the necessary iteration process in terms of displacement is carried out using the design normal force N_d. The analysis is based on a frame system, such that axial stiffnesses, and anchor and strut inclinations are taken into consideration correctly.

Subsequent design is based on a comparison of stresses

$$\sigma_d \leq f_{y,k} / \gamma_M = f_{y,k} / 1{,}1 = f_{y,d}$$

The 'Example' folder contains 4 **GGU-RETAIN** files, which deal with the classical Euler cases 1 to 4. If the vertical load V is increased slightly using the "**Editor 2/Action boundary conditions**" menu item and the system analysed, the following error message appears:



The normal force given in the data files thus corresponds to the buckling force determined after Euler.

7.19 Bedding conditions of wall toe

Classical methods of computation recognise two limit states for bedding of the wall toe:

- **Limit state 1:**
  Wall toe freely embedded on resultant of passive earth pressure.
  Embedment depth is determined through iterative displacement of a horizontal support.
  Embedment depth is found when support force is **zero**.
- **Limit state 2:**
  Wall toe fully fixed.
  Determination of the embedment depth is achieved through iterative displacement of a fixation point at the bottom of the system. The embedded depth is found when the moment is **zero** at the point of fixation.

For full fixation in accordance with the EAB, **GGU-RETAIN** increases the theoretical embedment depth by 20%. Between the two limit states, intermediate conditions of partial fixation between 0 % and 100 % are possible. If you want to compute partial fixations, **GGU-RETAIN** will first calculate embedment depths for the two limit states. The necessary embedment depth is then obtained through linear interpolation between the two extremes. In the same way, for a given section length, the degree of fixation is determined from the two extreme embedment depths. The necessary longitudinal extension (length addition) of between 0 % (for free earth support) and 20% (for complete fixation) is also determined by linear interpolation in compliance with the EAB.

Thus, there are two different ways in which section length and the degree of fixation can be determined:

- determination of section length for a given degree of fixation of wall toe;
- determination of degree of fixation of wall toe for a given section length.

7.20 Bedded systems

7.20.1 General information on types

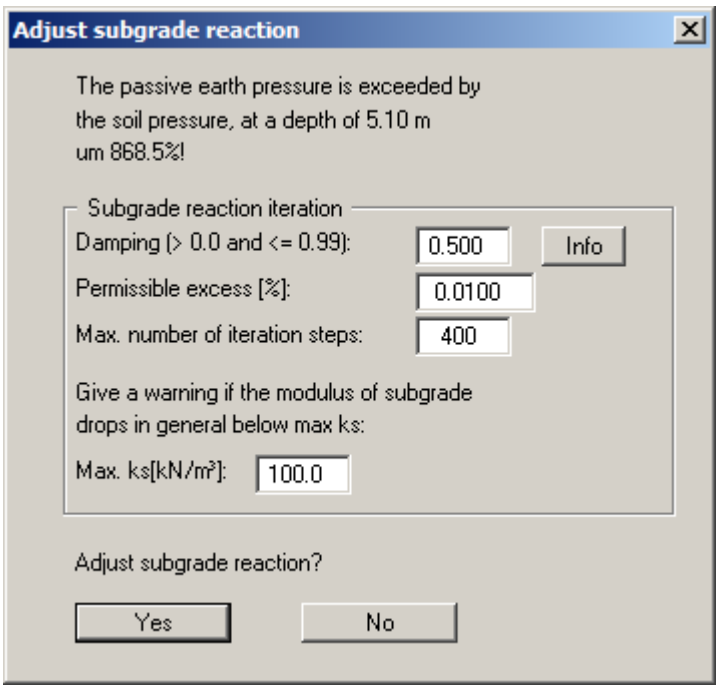
When a retaining wall is bedded there is always a degree of toe fixation, whereby the degree of fixation cannot be meaningfully given. Thus, for a bedded wall toe we have the following two variations:

- bedded wall toe with given section length;
- bedded wall toe with automatically determined section length.

7.20.2 Section length fixed and bedded toe

In this variation you enter a fixed height for the wall.

The elastic analysis produces a value for the foundation pressure that develops in front of the wall and which is determined from the product of the modulus of subgrade reaction and the deformation along the wall. This ***elastic pressure*** may not be greater than the passive earth pressure acting in front of the pile. Following analysis, this condition will generally not be met, so that iterative reduction of the modulus of subgrade reaction, along with the resulting soil pressure, will be necessary in order to meet it. The following dialog box will appear, with which you can control the iteration process.



The default values have proven themselves in practice and will generally not need to be altered.

"**Damping**" prevents strong fluctuations around the actual solution during the iteration process. A value of **0.0** produces no damping, while a value of **0.99** produces very strong damping and long computation times.

Using "**Permissible excess**" you specify the percentage by which the elastic stress may exceed the passive earth pressure before the iteration is ended.

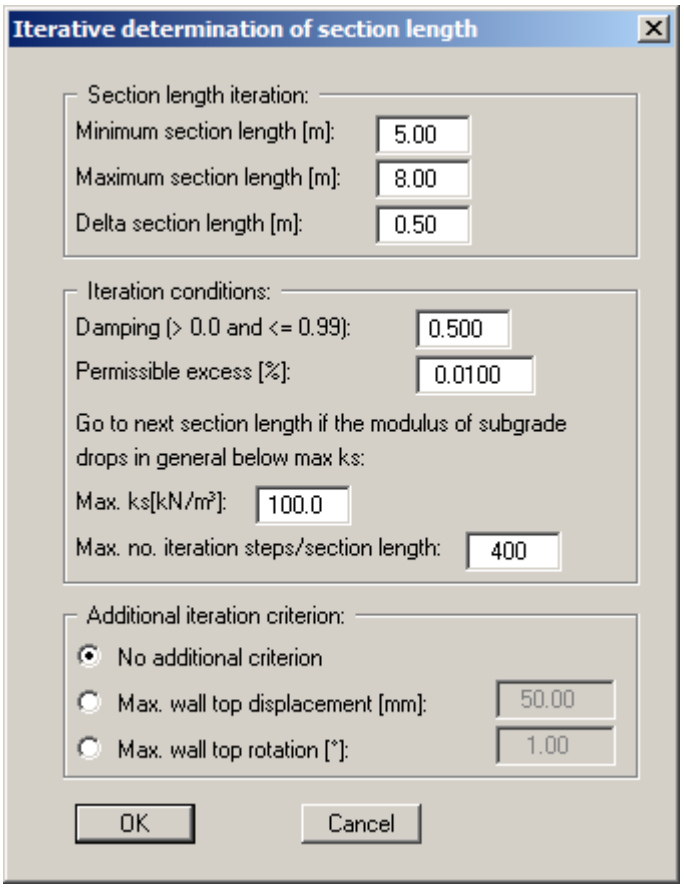
The number of iteration steps can also be specified. If, after the set number of steps, soil pressure is still greater than passive earth pressure, iteration is ended without result.

During iteration the modulus of subgrade reaction is reduced. If the analysis requires a reduction over the entire wall length to a value below "**max ks**", iteration is aborted. This usually happens when the wall is not long enough to take the load.

7.20.3 Automatically determined section length and bedded toe

Using this option you can determine the optimum height of the wall.

First you must define the range of section lengths within which **GGU-RETAIN** is to find the ideal value.



Enter the maximum and minimum section lengths, as well as a delta section length, in the top group box of the dialog box shown above. This defines the range of lengths from which the optimum length should be selected. With the data entered in the above dialog box, for example, the lengths 5.00, 5.50, 6.00, ..., 8.00 will be investigated. The central group box deals with iteration (see explanations in Section 7.20.2). In the bottom group box you can define additional iteration criteria, such as maximum displacement of the wall top.

7.21 Action and displacement boundary conditions

GGU-RETAIN automatically sets the boundary conditions shown in Figure 19 Possible structural system (see Section 7.16). It is also possible to enter additional boundary conditions for any point on the wall. All six state variables can be defined as boundary conditions:

- displacement along x-axis;
- displacement along y-axis;
- rotation;
- horizontal force;
- vertical force;
- moment.

7.22 Predeformations

When a number of excavation phases are under study, deformations from a previous phase can be implemented as lowering of supports at the new anchor points for the next phase. According to the EAB (R 11), however, such studies are not usually necessary.

In a continuous beam analysis, it is a simple matter to introduce the corresponding support displacements. In a rod framework application that does not consider anchors and struts as mere supports (see Figure 19 in Section 7.16), the same lowering of the support must be introduced at the anchor toe as at the anchor head, otherwise a large proportion of the resultant load would be transferred to the anchor. Thus, apart from being able to define displacement boundary conditions, **GGU-RETAIN** also offers a menu item dealing with predeformations. If you wish to enter predeformations by hand, or need to, this must be done as displacements under predeformations and not as displacement boundary conditions. Of course, once this is done, **GGU-RETAIN** takes care of the rest. You also have the option of automatically adopting predeformations from a previous analysis.

Designing a retaining wall involves a multitude of different adjustments, making it all too easy to lose one's bearings. **GGU-RETAIN** is thus designed to ask numerous questions, which, apart from controlling correct entry of data, also check plausibility, special preferences, and compatibility with the EAB. When a data entry error is recognised, an warning box appears and analysis is put on hold. Responses to inconsistencies in respect to plausibility or special preferences always offer two options:

- accept warning and start analysis;
- accept warning and do not start analysis.

When **GGU-RETAIN** draws your attention to special preferences , the dialog box is always designed so that simply pressing [**Return**] will not terminate the analysis.

GGU-RETAIN is designed for a high degree of user interaction, with the aim of avoiding erroneous and unwanted preferences (see above). Batch processing for the background processing of a number of files simultaneously is incompatible with this concept. Therefore, **GGU-RETAIN** does not create a single dataset when dealing with a number of construction phases. All phases must be dealt with in separate datasets. The only disadvantage of this is the treatment of predeformations. If you want predeformations from a previous construction phase to be automatically taken into consideration, this previous phase must already have been analysed (obviously) and be available as a dataset (on your hard disk). This dataset must be given as the dataset from which the predeformations are to be determined for the new construction phase. Since there is no re-analysis of the old phase, this dataset and the results obtained with it must be saved. If you should mistakenly select a dataset that contains no results, **GGU-RETAIN** will inform you of this. Otherwise, there are no restrictions. The name of the file with the predeformations is saved with the current dataset, where it is available for future use.

7.23 Prestressing

Personal experience has shown that discussions on the necessity of considering prestressing are never-ending. Regardless of these discussions, the program allows prestressing to be taken into consideration. Select the menu item "**Editor 2/Prestressing**".

Positive prestressing must be entered for anchors and struts. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts a force boundary condition equivalent in size and direction to the prestress into the system at the appropriate depth. Following analysis the prestress is added to the anchor force (strut force). This ensures that deformations only result from forces greater than the prestressing force.

7.24 Modulus of subgrade reaction

According to the EAB (R 11), elastic analysis with subgrade reaction moduli can be employed to determine internal forces. For this purpose, you can define linearly variable subgrade reaction modulus curves for the wall toe area, which can vary from area to area, making it possible to reproduce any subgrade reaction modulus curve you choose. Soil stresses result from analyses with subgrade reaction moduli, as the product of displacement and subgrade reaction moduli, which must not be greater than the passive earth pressure permitted at the point under consideration. Meeting this requirement generally requires iteration, which is performed by **GGU-RETAIN**.

The subgrade reaction modulus curve is shown on the screen, but only when one of the following two variations is selected (see Section 7.20):

- fixed section length and bedded toe;
- automatic determination of section length and bedded toe.

7.25 Earth pressure redistribution

Classical earth pressure redistribution is not appropriate for braced or tied-back retaining walls. Earth pressure calculated according to classical theory must be redistributed. **GGU-RETAIN** includes a large number of pre-programmed redistribution figures. In addition, by defining a polygon, the user can create any redistribution figure. The figures available are given below:

- rectangle;
- 2 rectangles;
- triangle, maximum at top;
- triangle, maximum at centre;
- triangle, maximum at base;
- trapezoid;
- rectangle with maximum at anchor locations or at any point;
- user-defined redistribution figure by means of a polygon;
- all EAB redistribution figures;
- EAU 2012.

For ***normal*** retaining walls, earth pressure redistribution according to the EAB is probably the simplest way of finding the correct redistribution figure. If you select this option, you only need to wait for the analysis to be completed. **GGU-RETAIN** selects the most appropriate from the 18 available EAB figures (R 69 for soldier pile walls and R 71 for sheet pile or in-situ concrete walls). If **GGU-RETAIN** is unable to find a suitable EAB figure because of ***abnormal*** anchor or strut positions, analysis is terminated and a warning message is issued. You must then give some thought to selecting or defining your own redistribution figure.

GGU-RETAIN gives you the option of including or excluding area loads in the redistribution. Water pressure is never redistributed.

> **Note:** In certain cases, according to the EAB R 13 and R 17, earth pressure redistribution can be analysed using a rectangular figure. However, increases in supporting forces and shear forces are then necessary. If, under earth pressure redistribution, you select "**Rectangle**" these increases are not implemented.

7.26 Base heave safety

Safety against base heave (see EAB R 10; Paragraph 1) is demonstrated using a bearing capacity failure analysis after Weißenbach (*Baugruben III, 1977*), which compares weight forces (including area loads, berms etc.) with the bearing capacity load, whereby a total of 50 vertical failure planes are investigated. The first failure plane is 0.2 times the excavation depth behind the retaining wall, the last failure plane is 5.0 times the excavation depth behind the retaining wall. For soldier pile walls, the bottom edge of the failure plane is the excavation bottom, while for sheet pile and in-situ concrete walls it is the base of the section.

Analysis of the safety against base heave of ***heavy foundations*** in the vicinity of excavations (see EAB R 10, Paragraph 2) is not performed.

7.27 General stability

The general stability (see EAB, R 10, Para. 3) can be simply verified by exporting the data from **GGU-RETAIN** to **GGU-STABILITY** (GGU slope stability application).

7.28 Hydraulic heave

7.28.1 Hydraulic heave safety using global safety factors

The hydraulic heave safety for each layer below the excavation base is determined via a comparison of the soil weights to the flow forces at the respective layer bases.

$$\eta_N = \frac{\sum_{i=1}^{N} G'_i}{S_N}$$

- η_N = hydraulic heave safety of the layer N
- G'_i = buoyant self-weight of layer i
- S_N = flow force of layer N
- layer 1 (i = 1) is the uppermost layer

The minimum value of all η_N is the hydraulic heave safety of the system.

7.28.2 Utilisation factor (hydraulic heave) using partial safety factors

Using the partial safety factor concept the following must be verified:

$$S'_k \cdot \gamma_H \leq \sum_{i=1}^{N} G'^i_k \cdot \gamma_{g,stb}$$

- S'_k = characteristic flow force on the percolated soil mass
- γ_H = partial factor for the flow force in favourable or unfavourable subsoil in the HYD limit state
- G'_k = characteristic dead load of the buoyant percolated soil mass
- $\gamma_{g,stb}$ = partial factor for stabilising permanent actions in the HYD limit state
- layer 1 (i = 1) is the uppermost layer

The so-called utilisation factor μ can also be calculated from this relationship.

$$\mu_N = \frac{S'_k \cdot \gamma_H}{\sum_{i=1}^{N} G'^i_k \cdot \gamma_{g,stb}}$$

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.28.3 Analysis of hydraulic heave after Aulbach/Ziegler

In 2013, Aulbach performed extensive investigations and derived a relationship for homogeneous ground conditions (Aulbach, Benjamin: *Hydraulischer Grundbruch - Zur erforderlichen Einbindetiefe bei Baugruben in nichtbindigem Baugrund*, dissertation, RWTH Aachen, 2013, URN: urn:nbn:de:hbz:82-opus-46909). The hydraulic heave safety can thus be directly determined.

The relationship for the three-dimensional case is shown below (abstract from Aulbach & Ziegler (2013)[1]):

$$\frac{T}{H} = Be \cdot \left[0{,}32 \cdot A + (1{,}244 - 0{,}32 \cdot A) \cdot e^{\left(\frac{\frac{B}{H}}{U \cdot \left(0{,}541 + 0{,}395 \cdot \left(1 - e^{\left(1 - \frac{S}{H}\right)}\right)\right) \cdot \left(1 + \left(\frac{B}{L} - 0{,}3\right) \cdot (3{,}156 - 1{,}564 \cdot U)\right)} \right)} \right] \cdot \left(\frac{\gamma'_{ref}}{\gamma_i \cdot 0{,}902 + 1{,}078} \cdot \frac{\eta_i}{\eta_{ref}} \right)^{\sqrt{2}}$$

Equation (6) in Aulbach & Ziegler (2013)[1] applies for the plane case:

$$\frac{T}{H} = 0{,}32 \cdot A + (1{,}244 - 0{,}32 \cdot A) \cdot e^{\left(\frac{\frac{B}{H}}{U \cdot \left(0{,}541 + 0{,}395 \cdot \left(1 - e^{\left(1 - \frac{S}{H}\right)}\right)\right)} \right)}$$

The result of the analysis is the necessary embedment depth.

[1] Aulbach, Benjamin; Ziegler, Martin:
Simplified design of excavation support and shafts for safety against hydraulic heave = *Einfache Bemessung von Baugruben und Schächten im Hinblick auf die Sicherheit gegen hydraulischen Grundbruch.*
In: Geomechnics and Tunnelling 6 (2013), No. 4, p. 362-374, ISSN 1865-7362.

7.29.1 Buoyancy safety using global safety factors

The buoyancy safety for each soil layer within the excavation is determined via a comparison of the soil weights to the water pressures at the respective layer bases. The self-weights of site plant and structures, frictional forces etc. are not included.

$$\eta_N = \frac{\sum_{i=1}^{N} G_i}{P_N}$$

- η_N = buoyancy safety of layer N
- G_i = self-weight of layer i
- P_N = water pressure at base of layer N
- layer 1 (i = 1) is the uppermost layer

The minimum value of all η_N is the buoyancy safety of the system.

If the same permeability has been defined for the whole system, safety against buoyancy is not determined. In certain cases, for example, when permeability on the passive side is much greater than on the active side, the calculation of safety factors for hydraulic uplift is meaningless. If, in such, or similar cases, the message "**Buoyancy safety could not be demonstrated**" appears, you can either ignore it or set the "**Safety against buoyancy**" to **1.0**, which suppresses the message.

7.29.2 Utilisation factor (buoyancy) using partial safety factors

Using the partial safety factor concept the following must be verified:

$$A_k \cdot \gamma_{g,dst} \leq \sum_{i=1}^{N} G_{k,stb}^{i} \cdot \gamma_{g,stb}$$

- A_k = the characteristic hydrostatic buoyant force acting on the lower surface of the complete structure, the soil layer in question or the excavation structure
- $\gamma_{g,dst}$ = partial factor for destabilising permanent actions in the UPL limit state
- $G_{k,stb}$ = lower characteristic value of stabilising permanent actions
- $\gamma_{g,stb}$ = partial factor for stabilising permanent actions in the UPL limit state
- layer 1 (i = 1) is the uppermost layer

The so-called utilisation factor μ can also be calculated from this relationship.

$$\mu_N = \frac{A_k \cdot \gamma_{g,dst}}{\sum_{i=1}^{N} G_{k,stb}^{i} \cdot \gamma_{g,stb}}$$

- μ_N = utilisation factor of layer N

Utilisation factors ≤ 1.0 mean that sufficient safety is given.

7.30 Analysis of sum H

When analysing the stability of soldier pile walls, the active earth pressure beneath the excavation base may be neglected (see EAB, R 15). However, it must be demonstrated that this neglected active pressure can be taken up with a sufficient margin of safety (generally 1.5) by the passive earth pressure. **GGU-RETAIN** performs the necessary investigation. It is possible that to comply with the EAB, R 15 an increase in embedment depth is necessary, which **GGU-RETAIN** will carry out automatically. If it is not possible to demonstrate wall stability, you are given advice on how to proceed according to the EAB.

7.31 Analysis of mobilised passive earth pressure

7.31.1 General note

The equilibrium of vertical forces must be verified in accordance with the EAB, R 9. **GGU-RETAIN** provides the necessary geostatic constants. to EABto EAB

7.31.2 Analysis of mobilised passive earth pressure using global safety factors

In accordance with the EAB R 9 and Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001), two cases must be distinguished:

- **Case 1**
  The vertical forces acting downwards are relatively small.
- **Case 2**
  The vertical forces acting downwards are greater than the vertical components of passive earth pressure determined using a negative angle of wall friction.

In Case 1, Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001) gives the following relationship:

$$\eta_v = (Pv + Eav + Cv + G) / Epv$$

Eav = vertical component of active earth pressure
Cv = vertical component of the counterforce C for toe fixation
G = self-weight of wall
Pv = permanent external loads (including vertical components of anchors)
Epv = vertical component of passive earth pressure

In accordance with the EAB, Cv is always determined as $^1/_3 \cdot \varphi$. The EAB gives a safety factor of 1.5.

In Case 2, Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001) gives the following relationship:

$$\eta_v = Qg / (Pv + Eav + G)$$

Qg = vertical ultimate bearing capacity of the wall or soldier piles
Qg can be determined pursuant to DIN 4026 or DIN 4014.

Further useful information relating to verification of Σ V can be found in Weißenbach (Geotechnical Engineering Handbook, Volume 3, 2001).

If the wall toe is bedded, Epv is determined from the positive subgrade reactions and the angle of wall friction.

7.31.3 Analysis of mobilised passive earth pressure using partial safety factors

Detailed notes on this analysis concept are included in EAU 2012, Section 8.2.5.5, and in the Recommendations on Piling (2012).

7.32 Analysis of vertical capacity

Detailed notes on this analysis concept are included in EAU 2012, Section 8.2.5.5, and in the Recommendations on Piling (2012).

7.33 Analysis of deep-seated stability

According to the EAB, R 44, the deep-seated stability of tied-back excavations must be verified. This primarily serves to determine the necessary anchor lengths. Verification uses the method described by Ranke/Ostermayer (*Bautechnik* 1968 (Construction Engineering), Issue 10). The example given in this article is provided as a file with this program (RANKE-Ostermeyer-d.VRB). When ***verifying deep-seated stability*** each anchor is first investigated (including the influence of the remaining anchors on the slip plane). Compound slip planes, which are determined by connecting the end points of the anchors involved, are then analysed.

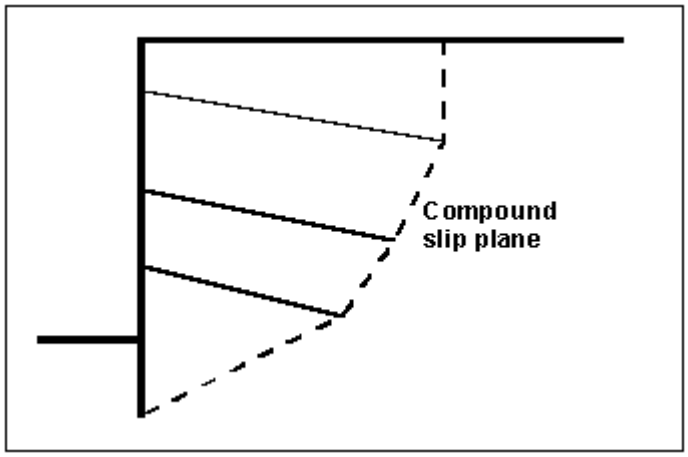


Figure 23 Compound "deep slip planes"

All possible combinations are analysed. For example, when there are four anchors:

- Slip plane passes through anchor end points
  1,2 and 1,3 and 1,4 and 1,2,3 and 1,2,4 and 1,3,4 and 1,2,3,4 and 2,3 and 2,4 and 2,3,4 and 3,4.

The only condition is that the next anchor end point is always to the right of and above the preceding one.

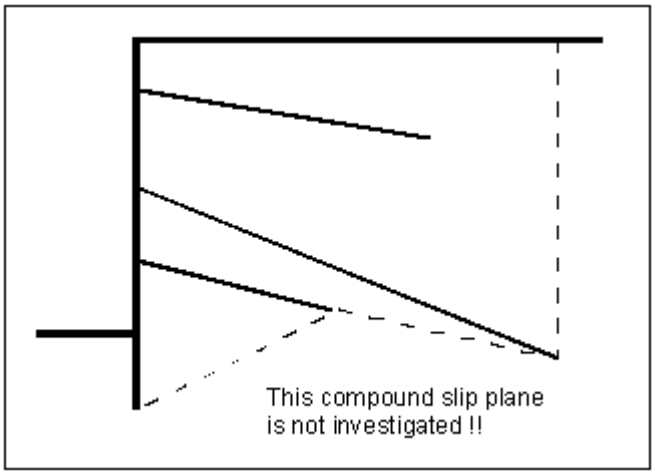


Figure 24 Compound "deep slip plane", which is not investigated

These slip planes are not critical. The most unfavourable slip plane associated with each anchor is displayed on the screen with the corresponding safety factor. A safety factor of 1.5 is generally required when adopting ***global safety factors***. If this safety factor cannot be achieved or is exceeded heavily, the program can optimise individual anchor lengths.

Using ***partial safety factors*** the possible anchor force is acquired in complete analogy to ***global safety factors***, but is divided by the passive earth pressure partial safety factor. The deep-seated stability is deemed as verified if:

$$A_{g,k} \cdot \gamma_g \leq A_{poss,k} \,/\, \gamma_{Ep} \ ,$$

where $A_{poss,k}$ is determined from the force polygon with permanent loads only, and

$$A_{g,k} \cdot \gamma_g + A_{q,k} \cdot \gamma_q \leq A_{poss,k} \,/\, \gamma_{Ep} \ ,$$

where $A_{poss,k}$ is determined from the force polygon with permanent and changeable loads. Where:

- $A_{g,k}$ = characteristic anchor force from permanent loads
- $A_{q,k}$ = characteristic anchor force from changeable loads

Here, too, optimisation with regard to a utilisation factor of 1.0 is possible.

7.34 Heave of anchor soil

Verification of heave of anchor soil is performed similar to the method described in Section 7.3.4 of the Piling Handbook 1977.

7.35 Construction phases

If you are investigating a number of advancing and retreating phases, **GGU-RETAIN** allows you to display the results in summarised form. For any selected phase it is possible to display the following conditions:

- envelope of moments;
- envelope of shear forces;
- envelope of normal forces;
- sum of deformations.

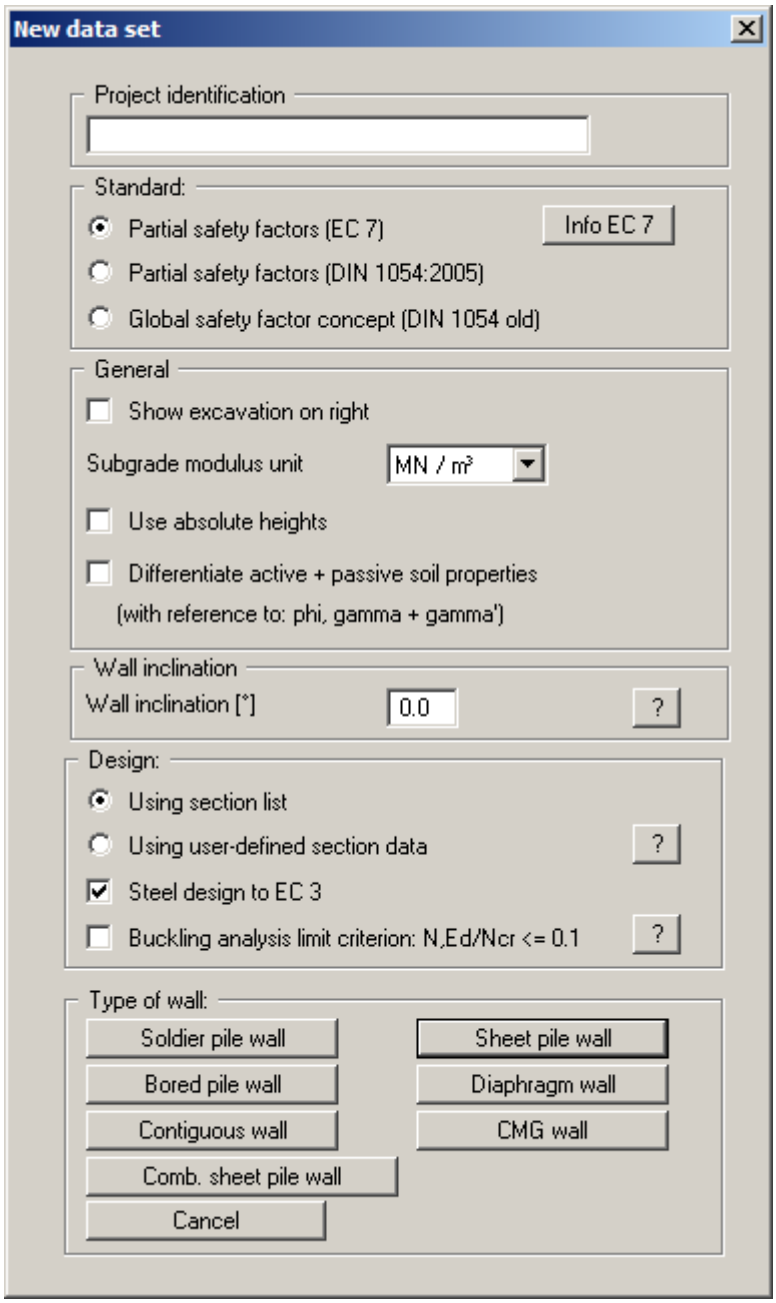
You can skip through the individual phases using the mouse.

8 Description of menu items

8.1 File menu

8.1.1 "New" menu item

You can enter a new system using this menu item. You will see the following dialog box:



You can enter a dataset description ("**Project identification**") of the problem going to process, which will then be used in the ***General legend*** (see Section 8.7.8). This can be particularly useful when you are working with the menu items "**Construction phases/Select files**" and "**Editor 2/Predeformation preferences**". Using these program functions previously saved datasets are appended to the current dataset. The existing project identifications are also shown in the appropriate dialog boxes to aid file characterisation.

In the next group box the radio buttons are used to specify which safety concept to use for analysis and design. Additionally, excavation visualisation to the right can be activated, as well as selecting kN/m³ or MN/m³ as the units for the modulus of subgrade reaction via a drop-down menu.

If you select the "**Use absolute heights**" check box you can enter all depths or heights in **m AD** (heights are positive upwards). If this check box is not selected, the top of the wall is assumed at **0.0** (height/depth) and all further entries, depth of soil layers, for example, are positive downwards. If, however, you want to work with absolute heights, enter the appropriate depths correspondingly altered. Thanks to ***WYSIWYG*** there is no danger of using incorrect data, since all input is immediately visible on the screen.

If your system uses differing soil properties on the active and the passive sides, activate the "**Differentiate active + passive soil properties**" check box in the above dialog box. You will then be presented with different input columns for entering the active and passive friction angle and unit weight soil properties in the "**Editor 1/Soils**" menu item (Section 8.2.5).

Below this, a wall inclination between -6° and +6° can be defined. Please read the information displayed after pressing the "**?**" button.

The "**Design:**" group box offers you the choice between a list of supplied sections (soldier pile sections, sheet pile sections, etc.) or user-defined sections for subsequent design. The user-defined sections offer the advantage of also being able to define different degrees of stiffness, moments of inertia, etc. for the retaining wall. The section list is loaded automatically when the program starts. Sections from a variety of well-known manufacturers can be accessed here. Clicking the "**?**" button will display further information. You can also use the "**File/New**" menu item to change from "**Using section list**" to "**Using user-defined section data**" for design. Steel design should always follow EC 3.

The buttons in the bottom section of the dialog box allow you to select the type of retaining wall you wish to analyse. Having selected a type of wall and entered all the data necessary for its analysis, you can subsequently return to this menu item and select another type of wall. **GGU-RETAIN** maintains the data already entered, which can be used to analyse the new type of wall.

If the safety concept has been altered using the "**File/New**" dialog box, the dialog box for specifying either the safety factors or the partial factors always opens after leaving the dialog box by clicking "**OK**". These dialog boxes can also be accessed via the menu items "**Editor 1/Verifications/Safety factors**" (see Section 8.2.13) or "**Editor 1/Verifications/Partial factors**" (see Section 8.2.14). Verifications and safety factors can be modified at any time using the above menu items.

8.1.2 "Load" menu item

You can load a file with system data, which was created and saved at a previous sitting, and then edit the system.

8.1.3 "Save" menu item

You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

8.1.4 "Save as" menu item

You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use "**.vrb**" as file suffix, as this is the suffix used in the file requester box for the menu item "**File/Load**". If you choose not to enter an extension when saving, "**.vrb**" will be used automatically.

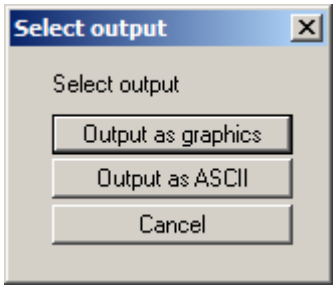
If the current system has been analysed at the time of saving, the analysis resultsare saved in the file. This is especially necessary if you want to use the results in this file for ***predeformations*** on other systems (see Section 7.22), or if you want to subsequently generate summaries of certain ***construction phases*** (see Section 7.35). If your current system represents an intermediate construction stage, it may be expedient to enter a further description of the file in "**Project identification**", in the "**File/New**" or "**Editor 1/Analysis options**" dialog box (see Section 8.1.1).

8.1.5 "Print output table" menu item

8.1.5.1 Selecting the output format

You can have a table printed containing the current analysis results. The results can be sent to the printer or to a file (e.g. for further editing in a word processor). The output contains all information on the current state of analysis, including the system data.

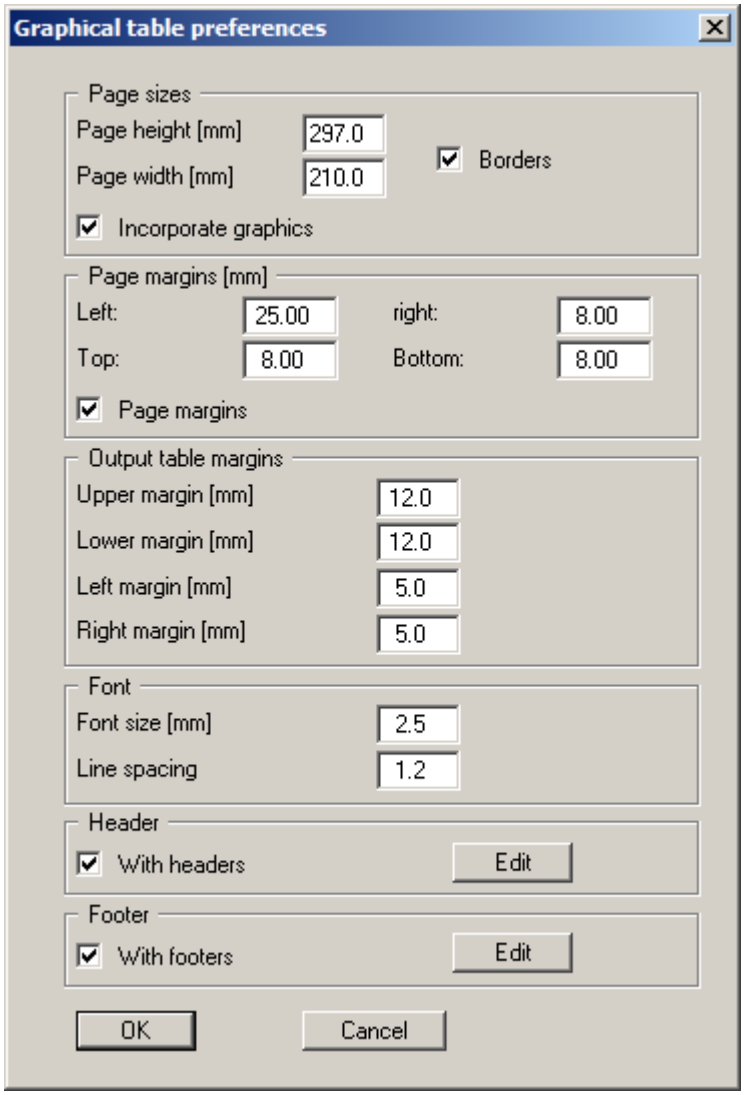
You have the option of designing and printing the output table as an annex to your report within the **GGU-RETAIN** program. To do this, select "**Output as graphics**" from the following options.



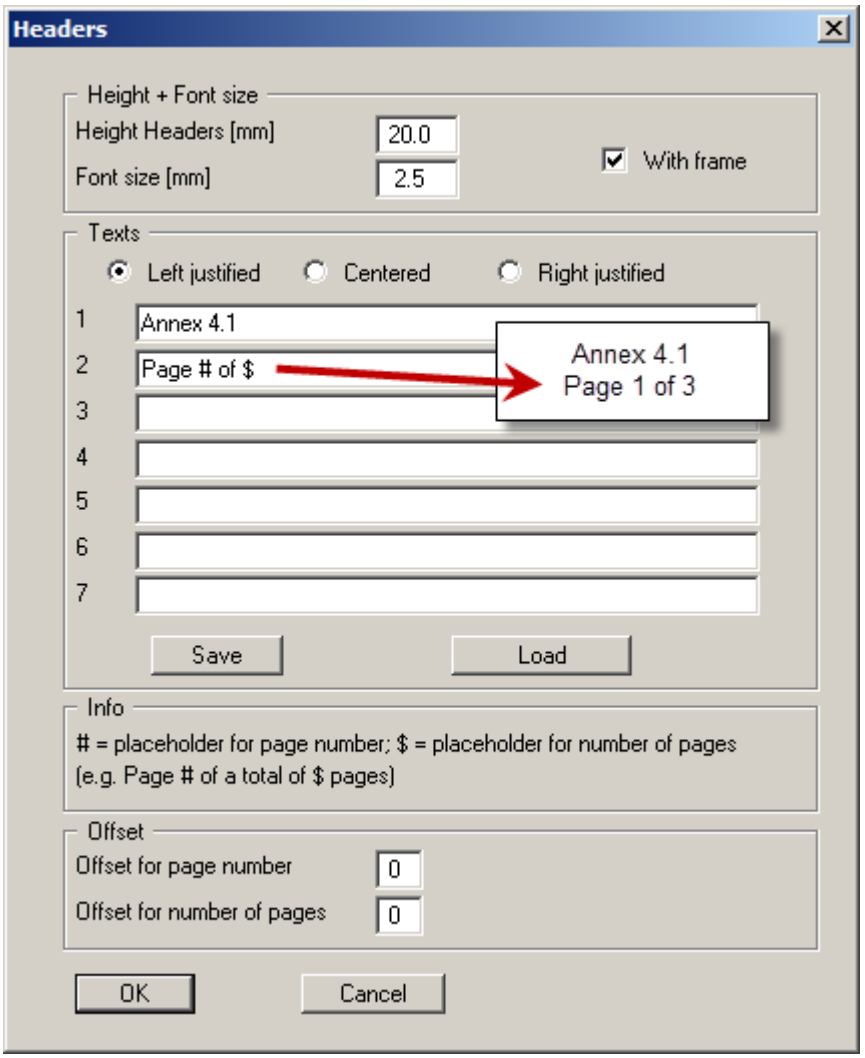
If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the "**Output as ASCII**" button.

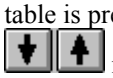
8.1.5.2 Button "Output as graphics"

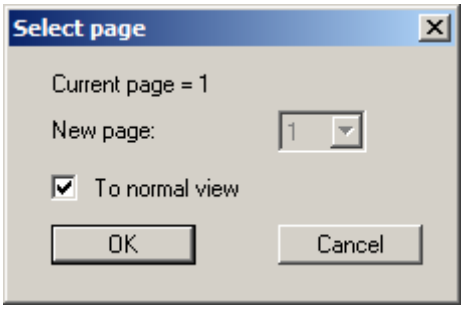
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box opens, in which you can define further preferences for result visualisation.



In the various group boxes of the dialog box, you can define preferences for the table output and layout. By activating the "**Incorporate graphics**" button, a sketch of the system is integrated in the output table. If you need to add a header or footer (e.g. for page numbering), activate the appropriate check boxes "**With headers**" and/or "**With footers**" and click on the "**Edit**" button. You can then edit as required in a further dialog box.

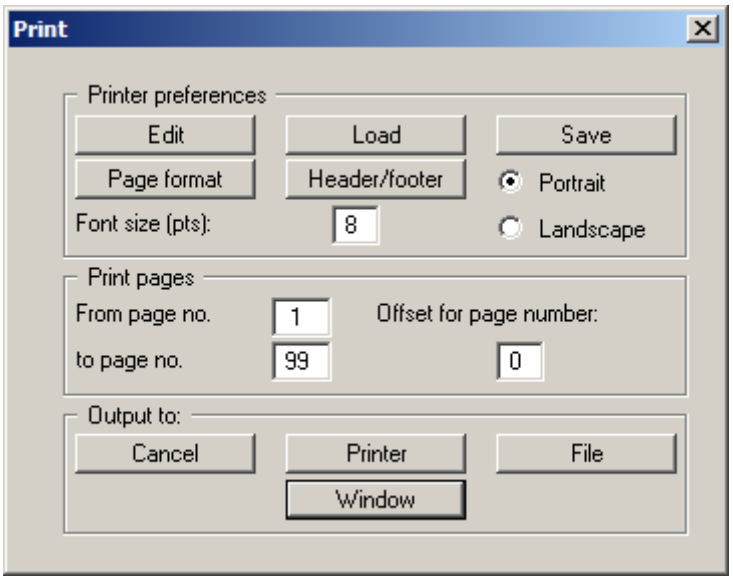


Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "**OK**" you will see a further dialog box in which you can select the parameters to be used in the output table. After confirmation the dialog box the output table is presented on the screen page by page. To navigate between the pages, use the arrow tools  in the toolbar. If you need to jump to a certain page or back to the graphical visualisation, click on the  tool. You will then see the following box:



8.1.5.3 Button "Output as ASCII"

You can have your analysis data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application.



In the dialog box you can define output preferences.

- "**Printer preferences**" group box
  Using the "**Edit**" button the current printer preferences can be changed or a different printer selected. Using the "**Save**" button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select "**GGU-RETAIN.drk**" as file name and save the file in the program folder (default), the file will be automatically loaded the next time you start the program.

  Using the "**Page format**" button you can define, amongst other things, the size of the left margin and the number of lines per page. The "**Header/footer**" button allows you to enter a header and footer text for each page. If the "#" symbol appears within the text, the current page number will be entered during printing (e.g. "***Page*** #"). The text size is given in "**Pts**". You can also change between "**Portrait**" and "**Landscape**" formats.

- "**Print pages**" group box
  If you do not wish pagination to begin with "***1***" you can add an ***offset number*** to the check box. This offset will be added to the current page number. The output range is defined using "**From page no.**" "**to page no.**".

- "**Output to:**" group box
  Start output by clicking on "**Printer**" or "**File**". The file name can then be selected from or entered into the box. If you select the "**Window**" button the results are sent to a separate window. Further text editing options are available in this window, as well as loading, saving and printing.

8.1.6 "Export" menu item

The general stability can be simply verified by exporting the data from **GGU-RETAIN** to **GGU-STABILITY** (GGU slope stability application). After clicking this menu item an appropriate file ("**.boe**") can be generated with the required **GGU-STABILITY** version status.

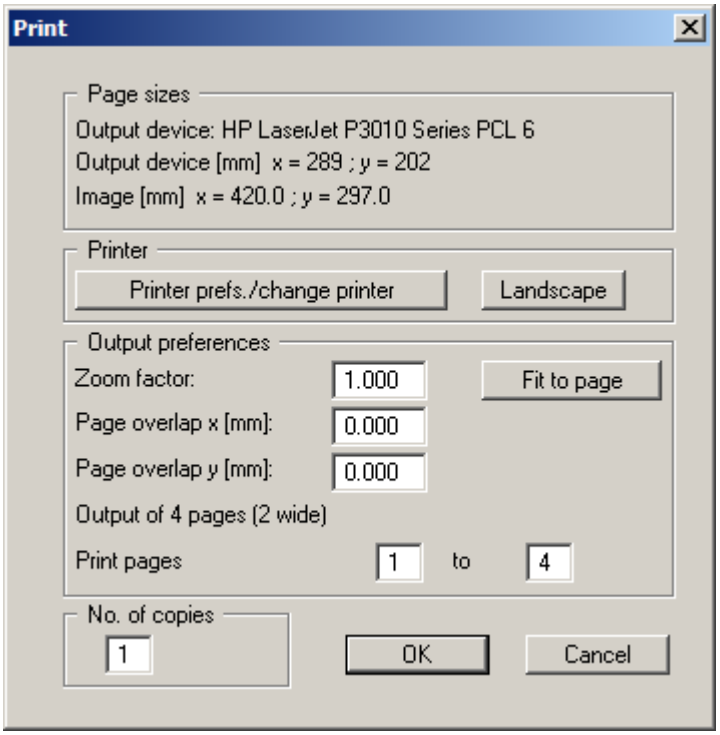
8.1.7 "Printer preferences" menu item

You can edit printer preferences (e.g. swap between portrait and landscape) or change the printer in accordance with WINDOWS conventions.

8.1.8 "Print and export" menu item

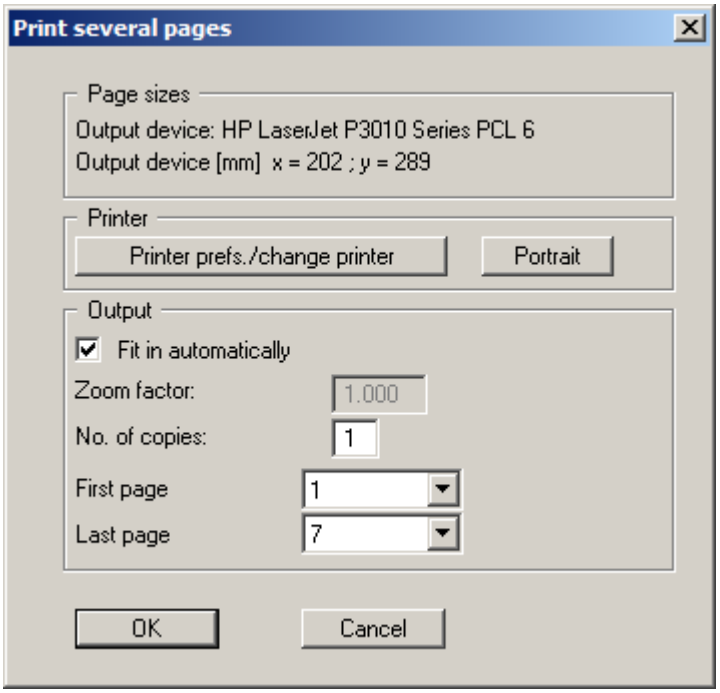
You can select your output format in a dialog box. You have the following options:

- "**Printer**"
  allows graphic output of the current screen contents (***graphical representation***) to the WINDOWS default printer or to any other printer selected using the menu item "**File/Printer preferences**". But you may also select a different printer in the following dialog box by pressing the "**Printer prefs./change printer**" button.



In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate better re-connection of the images, the possibility of entering an overlap for each page, in x and y direction, is given. Alternatively, you also have the possibility of selecting a smaller zoom factor, ensuring output to one page ("**Fit to page**" button). Following this, you can enlarge to the original format on a copying machine, to ensure true scaling. Furthermore, you may enter the number of copies to be printed.

If you have activated the ***table representation*** on the screen, you will see a different dialog box for output by means of the "**File/Print and export**" menu item button "**Printer**".



Here, you can select the table pages to be printed. In order to achieve output with a zoom factor of 1 (button "**Fit in automatically**" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "**File/Print output table**" button "**Output as graphics**".

- "**DXF file**"
  allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- "**GGUCAD file**"
  allows output of the graphics to a file, in order to enable further processing with the **GGUCAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.
- "**Clipboard**"
  The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. In order to import into any other WINDOWS program you must generally use the "*Edit/Paste*" function of the respective application.
- "**Metafile**"
  allows output of the graphics to a file in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

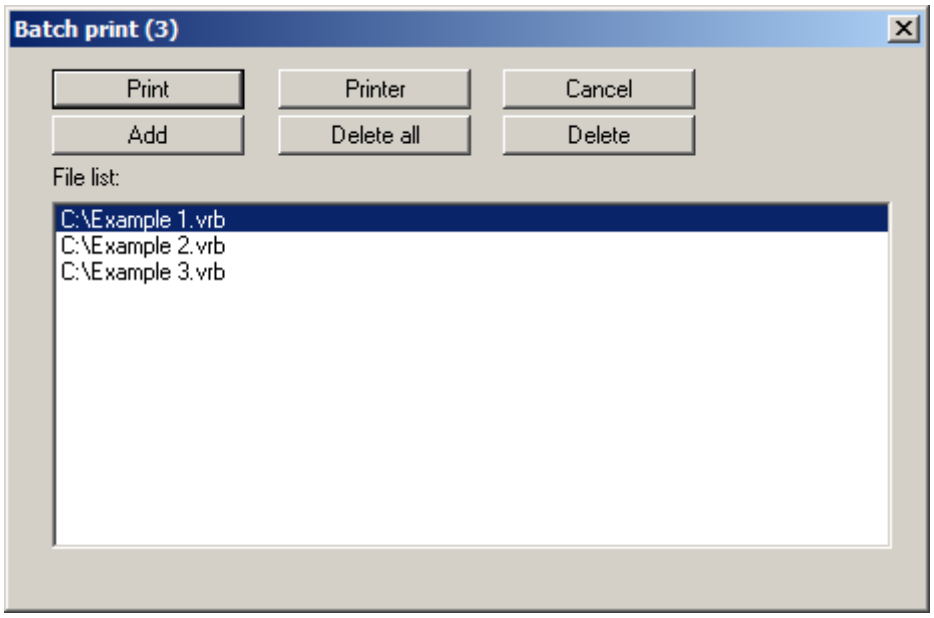
If you select the "**Copy/print area**" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 9.4).

Using the "**Mini-CAD**" program module you can also import EMF files generated using other GGU applications into your graphics.

- "**MiniCAD**"
  allows export of the graphics to a file in order to enable importing to different GGU applications with the **Mini-CAD** module.
- "**GGUMiniCAD**"
  allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program.
- "**Cancel**"
  Printing is cancelled.

8.1.9 "Batch print" menu item

If you would like to print several annexes at once, select this menu item. You will see the following dialog box:



Create a list of files for printing using "**Add**" and selecting the desired files. The number of files is displayed in the dialog box header. Using "**Delete**" you can mark and delete selected individual files from the list. After selecting the "**Delete all**" button, you can compile a new list. Selection of the desired printer and printer preferences is achieved by pressing the "**Printer**" button.

You then start printing by using the "**Print**" button. In the dialog box which then appears you can select further preferences for printer output such as, e.g., the number of copies. These preferences will be applied to all files in the list.

8.1.10 "Exit" menu item

After a confirmation prompt, you can quit the program.

8.1.11 "1, 2, 3, 4" menu items

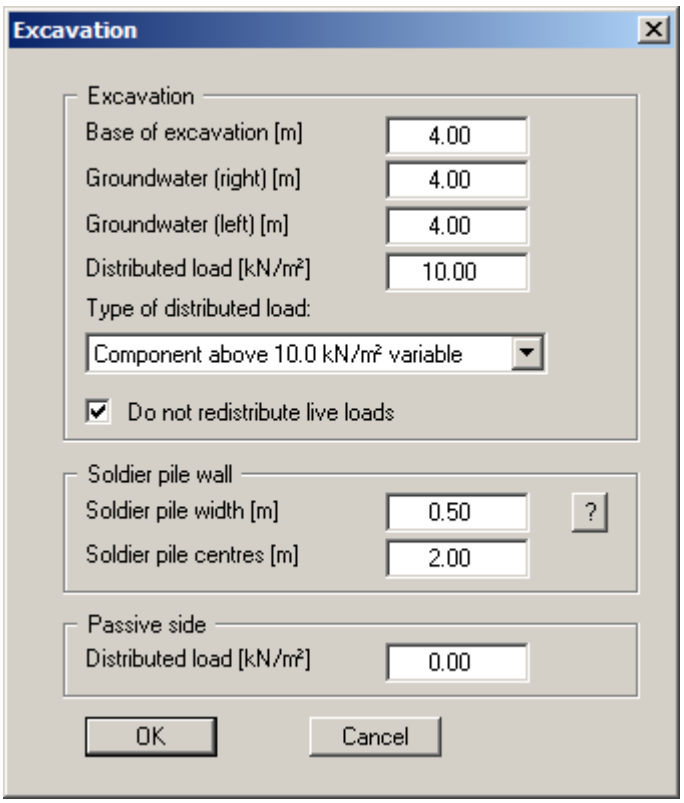
The "**1, 2, 3, 4**" menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous ***rummaging*** through various sub-folders.

8.2.1 "Analysis options" menu item

Using this menu item you can edit the default preferences of the current system. The dialog box corresponds to the box in the menu item "**File/New**" (see descriptions in Section 8.1.1).

8.2.2 "Excavation" menu item

A dialog box opens for defining the excavation and the selected retaining wall.

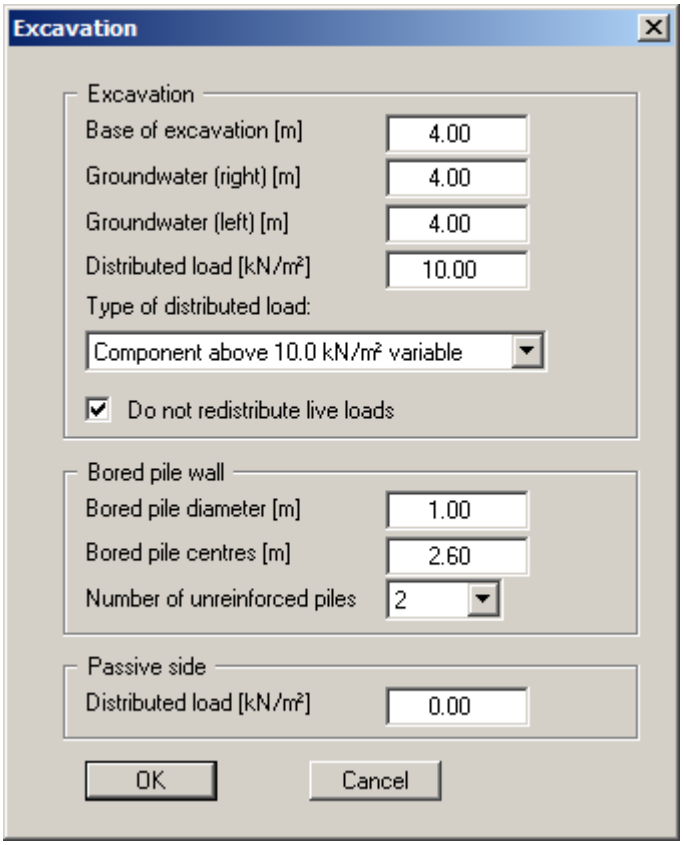


In the upper group box, enter the depth of the excavation base and the groundwater levels. If you checked the "**Use absolute heights**" box when defining the system, an additional entry, "**Top of wall**", appears in the dialog box for specifying the absolute position. In this case, all heights are measured in m AD or m site zero, i.e. the y-axis is positive upwards. You can then enter a value, for example, of 86.42 [m AD] in the "**Top of wall**" field. All further input must then be with reference to this value.

If the height of a previously defined system is subsequently set to absolute heights, a query follows after leaving the dialog box above asking for confirmation of whether soil strata and defined elements such as anchors, for example, should be adapted to the new wall top. Adaptation would mean that the depth of a soil layer entered as a positive value would be converted from, for example, 7.5 m to an absolute height of -7.5 m AD. If, then, you only convert your system to [m AD], do not select any elements in the query box and press the "**OK**" button.

Moreover, a distributed load can be defined. If you are working with the ***partial safety factors***, decide whether the distributed load is "**Permanent**", "**Changeable**" or the "**Component above 10.0 kN/m² changeable**" (see the following dialog box). "**Component above 10.0 kN/m² changeable**" means, for example, that for an input of 13.5 kN/m², 10 kN/m² are adopted as permanent and 3.5 kN/m² as changeable in the analysis (see EAB, R 7).

In the group box below this in the dialog box you make the required entries for the retaining wall. This section of the box varies depending on the type of wall and the chosen safety factor concept. For instance, to analyse a bored pile wall you may see the following box:



You can define the diameter, the centres of the reinforced bored piles and the number of unreinforced piles (secant pile wall). The above adaptations lead to the following bored pile wall:

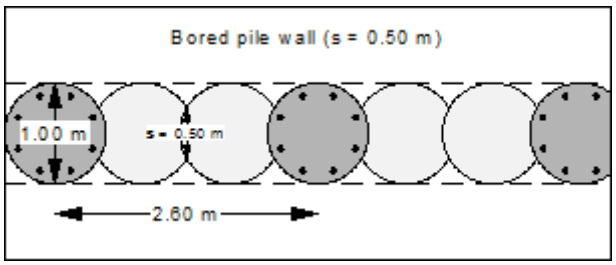
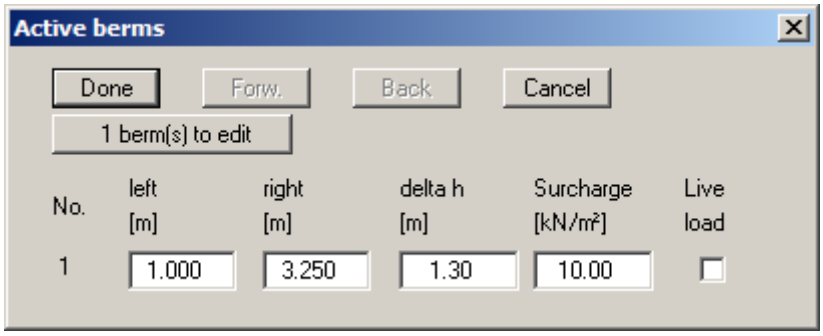


Figure 25 Dimensions of the bored pile wall

8.2.3 "Berms (active side)" menu item

You can define a maximum of 20 berms on the active side.



Enter the x-ordinates of the toe and head of the berm. With "**delta h**" you define the height of the berm, whereby negative values are also permitted. Finally, a "**Surcharge**" on the horizontal surface behind the head of the berm can be entered.

If more than one berm is present in the system, click "**x berm(s) to edit**" and enter the number of berms.

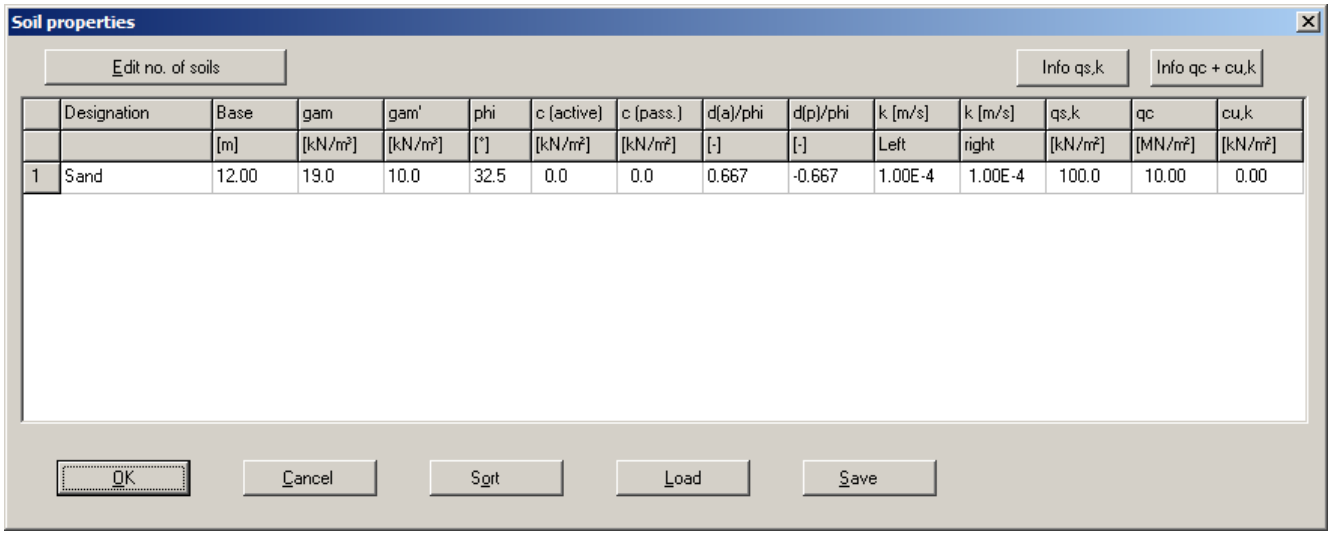
> Berms may not overlap. The program checks that this condition is adhered to and warns of any errors.

8.2.4 "Berms (passive side)" menu item

Berms on the passive side are defined in exactly the same manner as for the active side.

8.2.5 "Soils" menu item

You can define the soil properties in the following dialog box:



Layer depths (except when using the modulus of subgrade reaction) are always with reference to the top of the wall, or are absolute heights (m AD), if this was selected in the initial dialog box of the "**File/New**" menu item. If, when analysing a soldier pile wall after Weißenbach, you wish to make full use of passive earth pressure, enter a value of **-1.0**, for "**d(p)/phi**" (passive angle of wall friction/angle of friction). **GGU-RETAIN** automatically performs the reductions required by Weißenbach.

If you have activated the "**Differentiate active + passive soil properties**" check box in the dialog box in "**File/New**" or "**Editor 1/Analysis options**", you can enter differing friction angles and unit weights for the active and the passive sides. It is only necessary to enter permeabilities if hydrostatic pressures must be considered, and any hydraulic gradients, using flow conduits (see Section 7.7).

In addition, you can enter the soil parameter qs,k (= skin friction). However, this is only relevant if you want to analyse the pull-out resistance of anchors. Otherwise it is not taken into consideration and does not appear in the soil properties legend. If the check box "**Verification with qs,k**" in the "**Editor 2/Anchors**" dialog box is activated (see Section 8.3.7), the soil parameter qs,k is also shown in the soil properties legend.

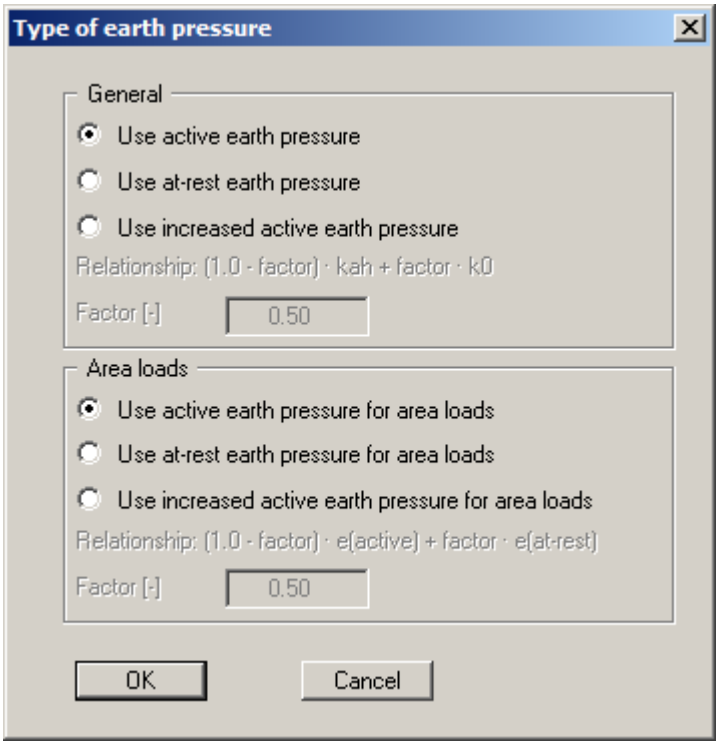
To analyse the vertical capacity to EAU, EAB and EAP, enter the cone resistance qc and the shear strength of the undrained soil cu,k.

In stratified soils the number of layers must be entered under "**Edit no. of soils**". Clicking the "**Sort**" button sorts the soil layers according to depth; however, this is performed automatically when you click "**OK**" to leave the dialog box. This eliminates the possibility of input errors.

> You can also use this function to eliminate a soil from the table.
> Simply assign the soil to be eliminated a greater layer depth and then click the "**Sort**" button. The corresponding soil is now the last soil in the table and can be ***deleted*** by reducing the number of soils.

8.2.6 "Type of earth pressure" menu item

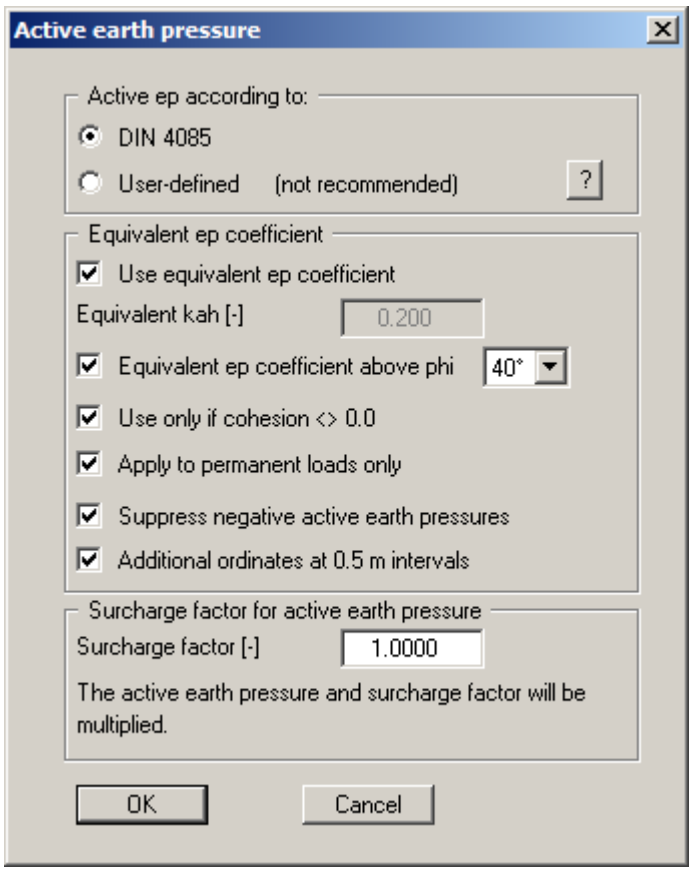
In this dialog box you define the type of earth pressure on which the analysis is to be based.



The options for ***area loads*** can be specified separately.

8.2.7 "Active earth pressure" menu item

You can specify active earth pressure preferences using this dialog box:



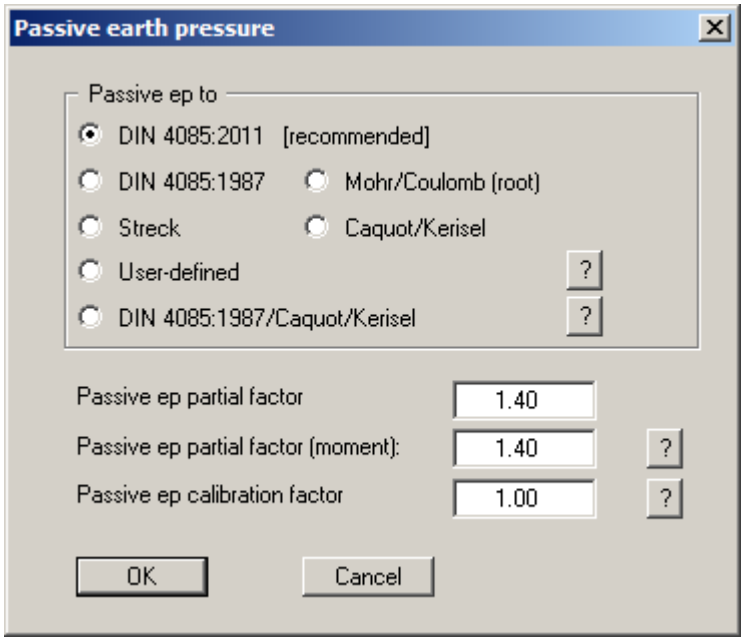
In the upper group box you specify the type of active earth pressure calculation. The method of choice is "**DIN 4085**". The other two methods are only of interest if you wish to analyse an example from older literature sources or check certain results. User-defined earth pressure coefficients can be entered using the menu item "**Editor 1/User-defined earth pressure coefficients**" (see Section 8.2.10).

The "**Use equivalent ep coefficient**" check box should only be deactivated in exceptional circumstances (see EAB R 4). The equivalent earth pressure coefficient can only be smaller than 0.2 in special circumstances (see EAB R 4). It only makes sense to deactivate this check box when re-examining existing analyses (for instance, all the examples used in the Piling Handbook). Alternatively, the equivalent earth pressure coefficient can be defined by means of a friction angle phi = 40° or phi = 45°. This procedure also takes the defined wall friction angle into consideration.

A number of applications on the market also provide the option of a general increase in active earth pressure, apart from certain forms or earth pressure redistribution. In order to be able to check calculations performed with such an application, **GGU-RETAIN** also offers this possibility.

8.2.8 "Passive earth pressure" menu item

You can specify passive earth pressure preferences using this dialog box:



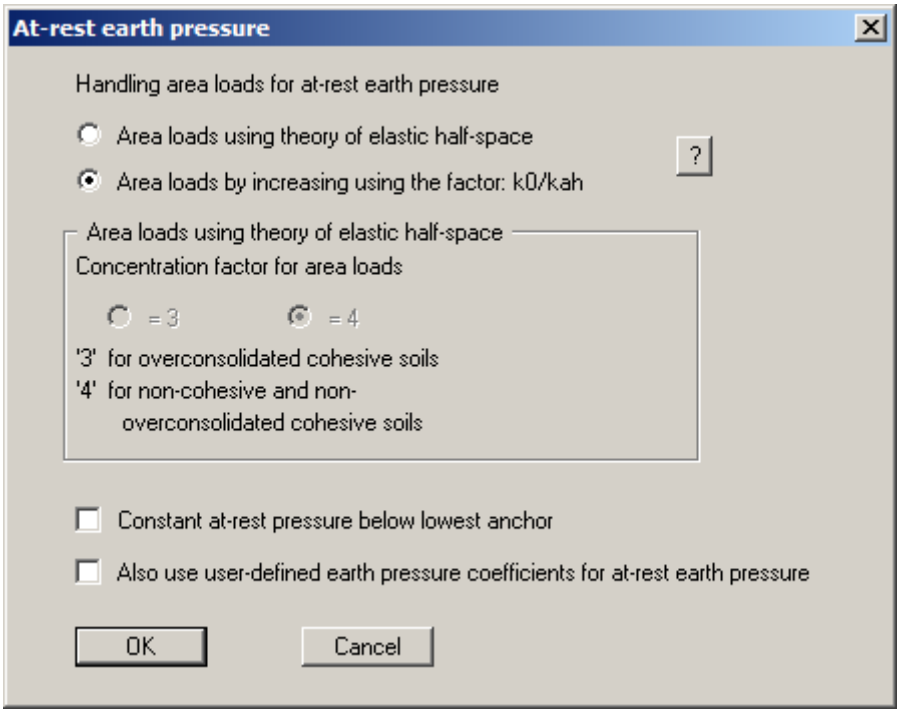
In the upper group box you specify the type of passive earth pressure calculation. The method of choice is "**DIN 4085:2011**". Only if you want to check examples from older literature or examine certain analysis results does it make sense to select a different method. User-defined earth pressure coefficients can be entered using the menu item "**Editor 1/User-defined earth pressure coefficients**" (see Section 8.2.10).

If you have selected ***partial safety factors***, enter the partial factor for passive earth pressure in the dialog box and the calibration factor in accordance with the information in DIN 1054:2010/EC 7. The partial factor for passive earth pressure used to calculate the design moment can be reduced according to EAU 2012 under certain boundary conditions.

8.2.9 "At-rest earth pressure" menu item

Using the at-rest earth pressure function, the area loads are calculated by increasing by the factor k0/kah in compliance with DIN 4085:2011 Section 6.4.3.

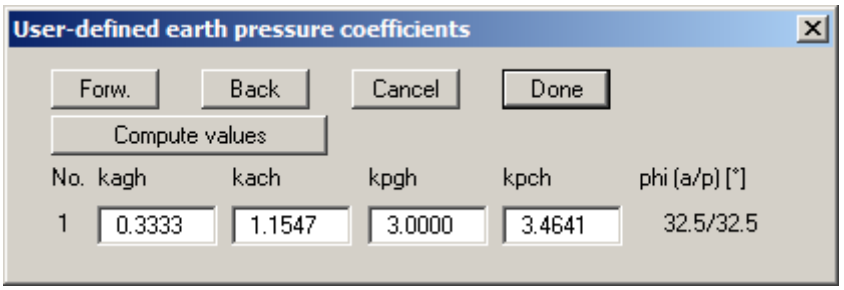
Alternatively, it is possible to determine the demands placed on the wall by area loads according to the theory of elastic half-space. The required concentration factor is specified in the following dialog box, if the upper check box is activated.



According to the EAB, R 23 (Paragraph 2), for retaining walls with at least two rows of anchors or struts, at-rest earth pressure can be kept constant from the lowest support. This is effected by selecting the check box at the bottom of the dialog box.

8.2.10 "User-defined earth pressure coefficients" menu item

If you want to work with user-defined earth pressure coefficients instead of with those computed by the program you can enter them here. Enter the coefficients for horizontal ground. If necessary, **GGU-RETAIN** will convert them for sloping ground using equation values for k_{ah0} und $k_{ah\beta}$ (see "**Theoretical principles/Berms**", Section 7.8).

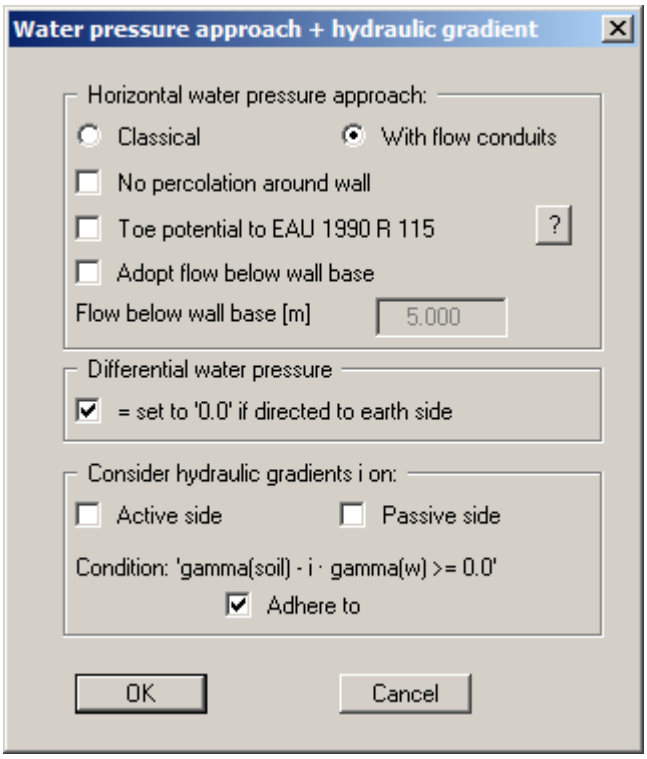


If the "**Compute values**" button is pressed the earth pressure coefficients can be calculated by the program in accordance with the required standard and slope angle.

The earth pressure coefficients entered here are only adopted if the "**User-defined**" check box is activated in "**Editor 1/Active earth pressure**" and/or in "**Editor 1/Passive earth pressure**" menu items (see Sections 8.2.7 and 8.2.8).

8.2.11 "Groundwater" menu item

With regard to adopting horizontal water pressure you are referred to the explanations in Section 7.7.

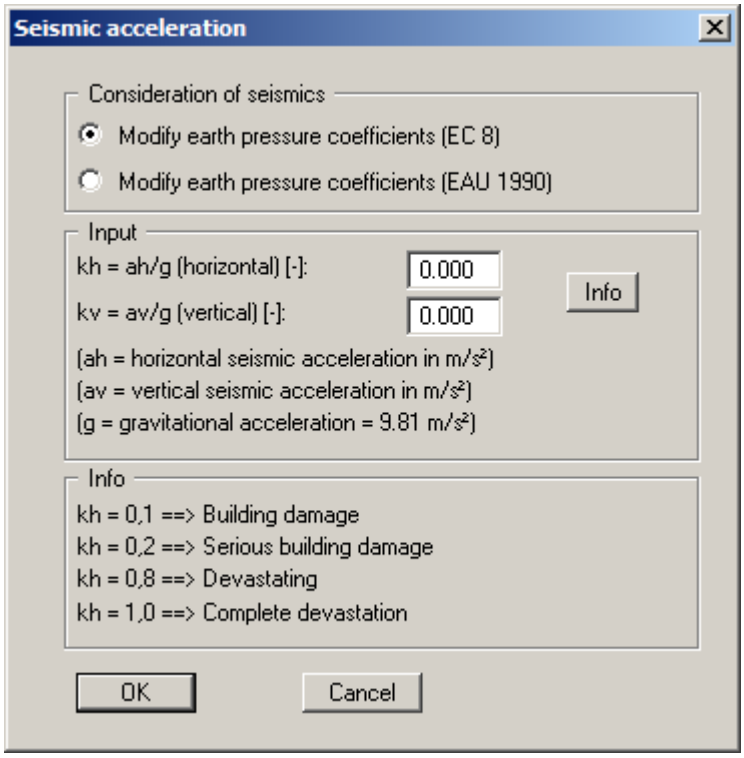


If the base of the sheet pile does not correspond to the base of the actual percolation (e.g. combined sheet pile wall with infill sheets not driven to the depth of the king pile), a depth may be defined for the base of the percolation below the toe after activating the check box "**Adopt flow below wall base**".

The retaining walls of an excavation standing in flowing groundwater will experience increased earth pressure on the active side and decreased earth pressure on the passive side. According to the EAB, R 63, the effect on the active side can generally be ignored, whilst the effect on the passive side must always be taken into consideration. You can make the appropriate selection in the bottom section of the dialog box.

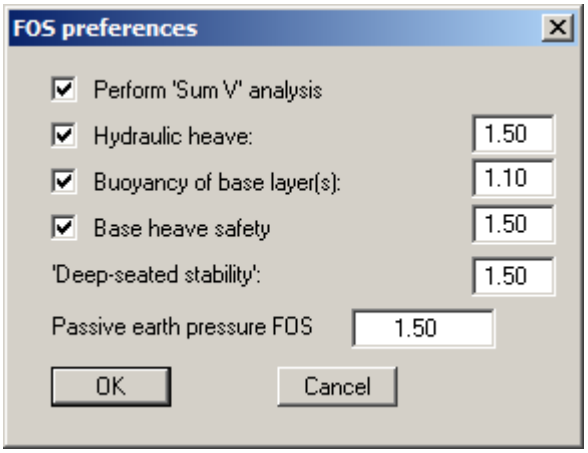
8.2.12 "Seismic acceleration" menu item

Seismic loads can be taken into consideration as described in EC 8 or EAU 1990, Section 2.14, by increasing the active earth pressure coefficients and reducing the passive earth pressure coefficients. Seismic loads are given in multiples of gravitational acceleration (g).



8.2.13 "Verifications/Safety factors" menu item

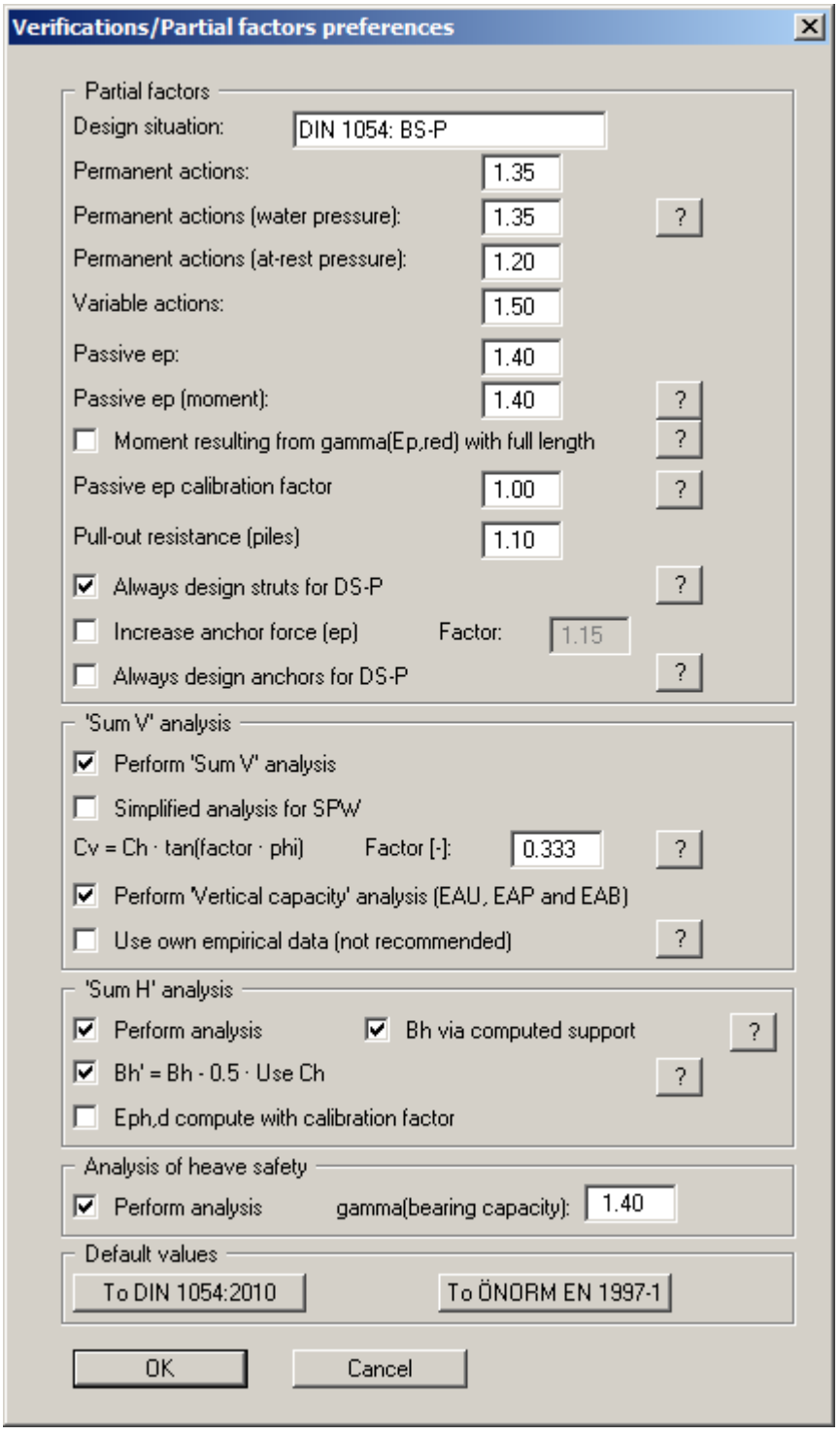
If you are analysing with ***global safety factors*** to DIN 1054 old, you can specify the verifications to be performed by the program by activating the corresponding check boxes in the dialog box shown below.



Then enter the safety factors for the individual verifications. The program's default values in the box are those required by the EAB. For soldier pile walls, the safety factors for "**Hydraulic heave**" and "**Buoyancy of base layer(s)**" are removed. Then, the verification of "**Sum H**" can be activated instead. If you do not want the program to perform one of the verifications, the corresponding check box must be deactivated.

8.2.14 "Verifications/Partial factors" menu item

If you are analysing with the ***partial safety factor concept*** , you will see a dialog box for defining the partial factors.



The partial factor for the permanent action as a result of water pressure can be reduced according to EAU 2012 under certain boundary conditions. The same applies to the passive earth pressure partial factor used for calculating the design moment.

In the dialog box you can decide whether to verify "**Sum V**" or "**Sum H**". The angle of the equivalent force C can be entered. The "**Sum H**" check box is only visible if you have selected "**Soldier pile wall**" or "**Contiguous wall**" as the retaining wall.

By pressing the "**To DIN 1054:2010**" button in the "**Default values**" group box the partial factors given in DIN 1054:2010 or EC 7 for the various load cases can be adopted. The load case designations were altered for the EC 7 partial safety factor concept:

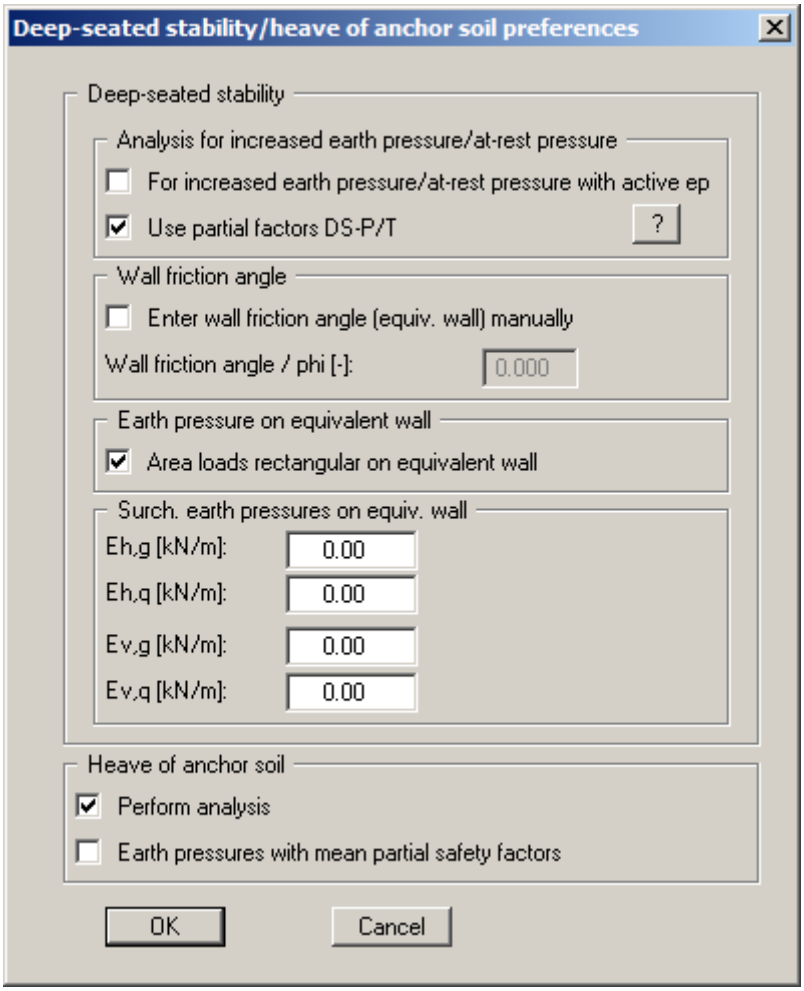
- Load Case 1 is now DS-P: Persistent Design Situation
- Load Case 2 is now DS-T: Transient Design Situation
- Load Case 3 is now DS-A: Accidental Design Situation

In addition, there is a seismic design situation (DS-E). In the DS-E design situation all partial factors = '**1,0**'.

In addition, it is possible to select the partial safety factors compliant with Austrian standards using the "**To ÖNORM EN 1997-1**" button.

8.2.15 "Deep-seated stability/Heave of anchor soil" menu item

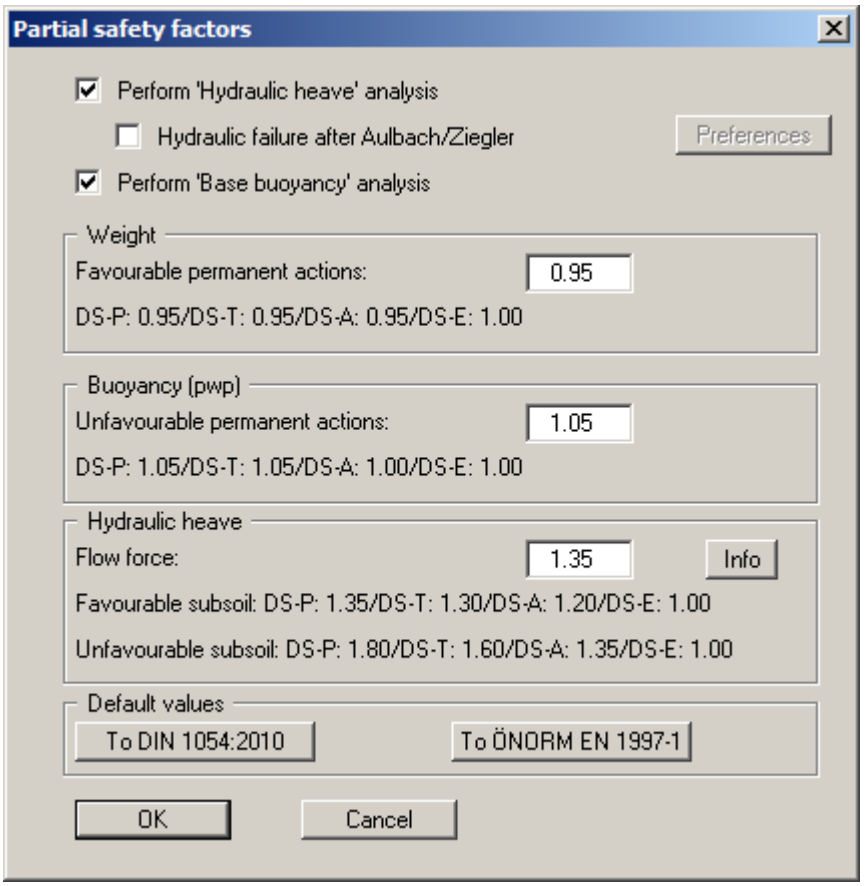
Use this menu item to define preferences for analysis of deep-seated stability.



Verification of ***heave of anchor soil*** can be activated in the lower group box of the dialog box. If a value unequal to '**0**' is entered for the height of the dead man when entering the anchors, and this check box is activated, verification is performed similar to the method described in Section 7.3.4 of the Piling Handbook 1977.

8.2.16 "Buoyancy + Hydraulic heave" menu item

This menu item opens a dialog box in which you can activate 'traditional' hydraulic heave analysis and analysis of the buoyancy of the excavation base. In addition, the hydraulic heave safety can be analysed using the Aulbach/Ziegler method (Aulbach, Benjamin; Ziegler, Martin: Simplified design of excavation support and shafts for safety against hydraulic heave = *Einfache Bemessung von Baugruben und Schächten im Hinblick auf die Sicherheit gegen hydraulischen Grundbruch*. In: Geomechanics and Tunnelling 6 (2013), No. 4, p. 362-374, ISSN 1865-7362.).



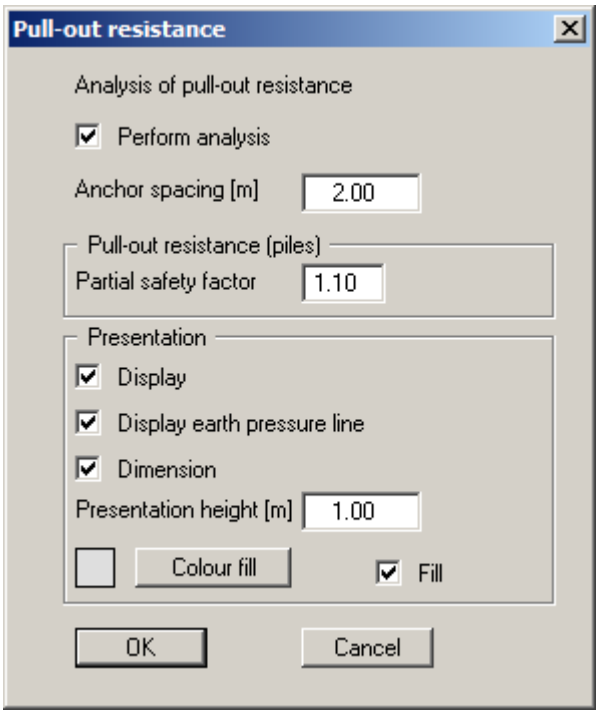
If you have activated "**Hydraulic failure after Aulbach/Ziegler**", you can enter the required system data via the "**Preferences**" button.

If you press the "**Info**" button, you will see further information on the subsoil situation (favourable/unfavourable). In the "**Default values**" group box the partial factors for the various load cases and subsoil conditions given in the DIN 1054:2010 and in the EC 7 can be selected by means of the dialog box reached by clicking the "**To DIN 1054:2010**" button.

In addition, it is possible to select the partial safety factors compliant with Austrian standards using the "**To ÖNORM EN 1997-1**" button.

8.2.17 "Pull-out resistance" menu item

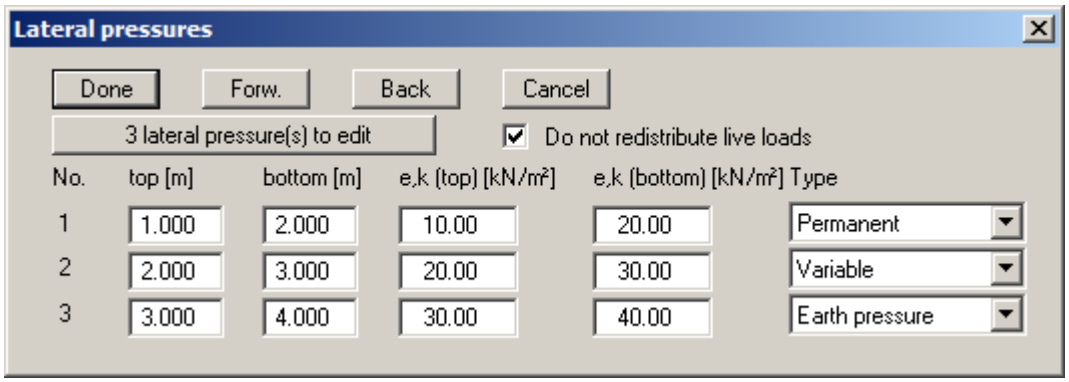
If anchors with activated "**Verification with qs,k**" check box are employed (see menu item "**Editor 2/Anchors**" in Section 8.3.7), the pull-out resistance analysis can be activated via this menu item.



Preferences for the following graphical visualisation can be specified in the lower group box of the dialog box.

8.3.1 "Lateral pressures" menu item

If, in addition to the diverse possibilities for determining earth pressure on the wall, you also need to take additional surcharges into consideration, this is where to enter them.

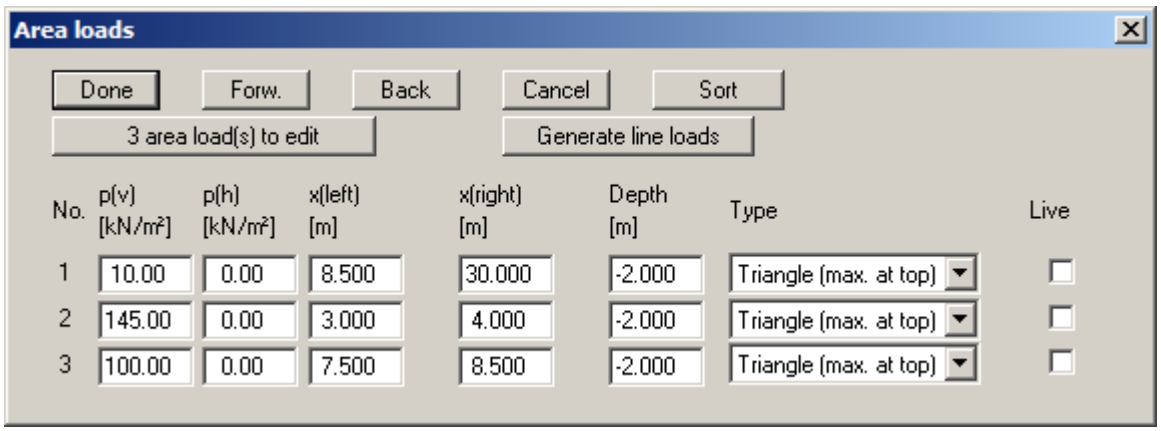


The number of lateral pressures can be modified using the "**x lateral pressure(s) to edit**" button**.** Then enter the ordinates in metres from the top of the wall or as absolute heights, and the values for the lateral pressures.

The lateral pressures can be represented by permanent or variable loads, water pressures or passive earth pressures. The characteristic values are entered and are automatically multiplied or divided by the appropriate partial safety factors.

8.3.2 "Area and line loads" menu item

Using this menu item you define area loads and line loads.

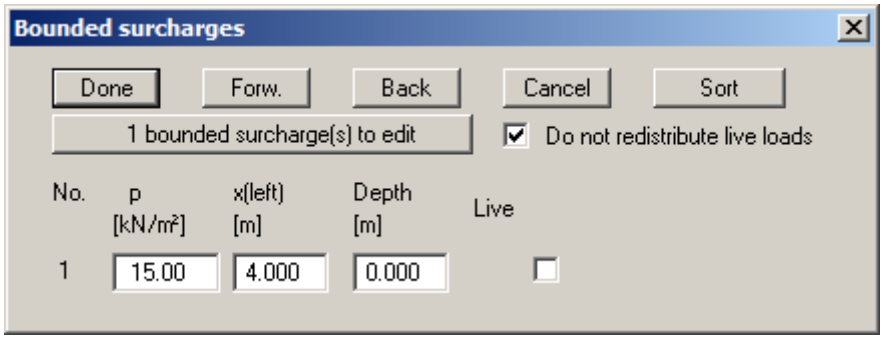


The "**x area load(s) to edit**" button allows you to determine the number of area loads to be considered. Subsequently you can enter the sizes "**p(v)**" (= vertical) and "**p(h)**" (= horizontal), the ordinates and the "**Depth**" of the area loads. You must also enter the "**Type**" (wall pressure shape) of the resultant horizontal forces on the wall (see "**Theoretical principles**", Section 7.9).

Using the "**Generate line loads**" button, line loads acting vertically on the wall may be considered adopted as area loads (see Section 7.10).

8.3.3 "Bounded surcharges" menu item

Bounded surcharges are defined in the following dialog box. You are first queried whether the surcharges are entered to be on the active or the passive side. The following dialog box opens for ***active side*** input:

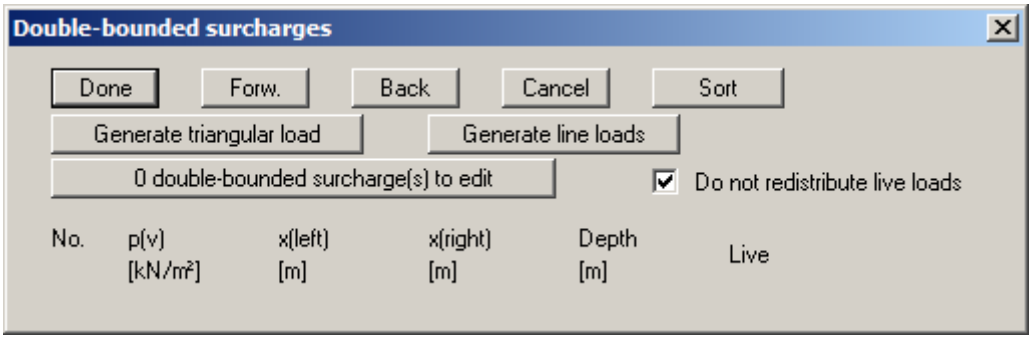


Using "**x bounded surcharge(s) to edit**" you can determine the number of bounded surcharges. You can then enter the size of the surcharges, the ordinates and the depth in metres from the top of the wall or as absolute heights.

Input of bounded surcharges on the ***passive side*** is analogous, but without the "**Live**" check box.

8.3.4 "Double-bounded surcharges" menu item

Double-bounded surcharges are defined using this menu item. You are first queried whether the surcharges are entered to be on the active or the passive side. The following dialog box opens for ***active side*** input:

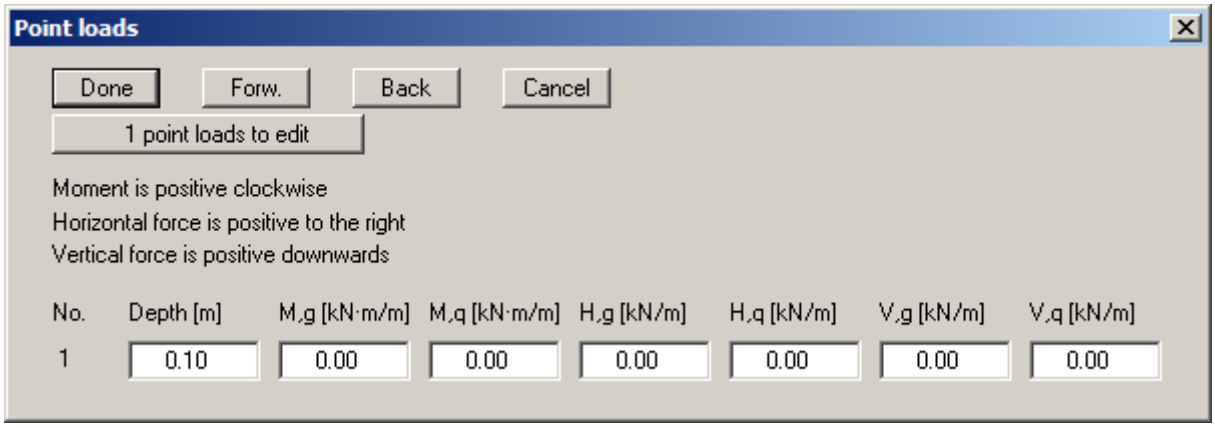


The number of surcharges can be edited using the "**x double-bounded surcharge(s) to edit**" button. Then enter the size of the surcharges, the ordinates and the depth in metres from the top of the wall or as absolute heights.

Input of bounded surcharges on the ***passive side*** is analogous, but without the "**Live**" check box.

8.3.5 "Action boundary conditions" menu item

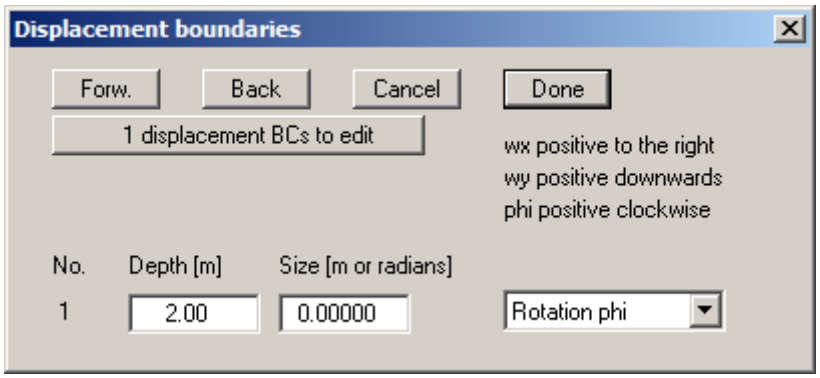
You can introduce additional action boundary conditions anywhere along the wall.



The direction of the forces is defined by means of the sign. In the example above a horizontal force of 15 kN/m has been entered at the top of the wall, acting towards the left. When adopting the ***partial safety factor concept*** the forces are subdivided into permanent and changeable forces.

8.3.6 "Displacement boundary conditions" menu item

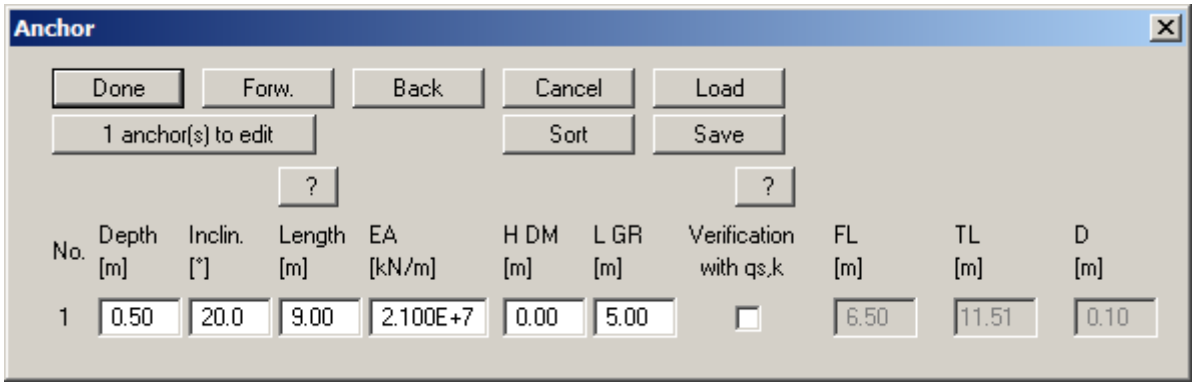
You can introduce displacement boundary conditions anywhere along the wall.



In the example above a "**Rotation phi**" of the wall of 0.0 has been entered 2.0 m below the top of the wall. The "**Displ. wx**" and "**Displ. wy**" switches stand for the horizontal and vertical displacements; the direction is defined by means of the sign.

8.3.7 "Anchors" menu item

Anchors are defined in the dialog box of this menu item.



It is necessary to enter the following information:

- Depth [m] = position of anchor head
- Inclination [°] of anchor
- Length [m] of anchor = starting point of deep-seated stability
- EA [kN/m] = axial stiffness of anchor steel
- H DM [m] = height of deadman (if present)
  This value is only necessary for analysis of deep-seated stability.
- L GR [m] = length of grouted section!!!.
  This value is of no importance for the analysis itself, but for the graphical visualisation.
  The length labelling of the anchor includes half the length of the grouted section.

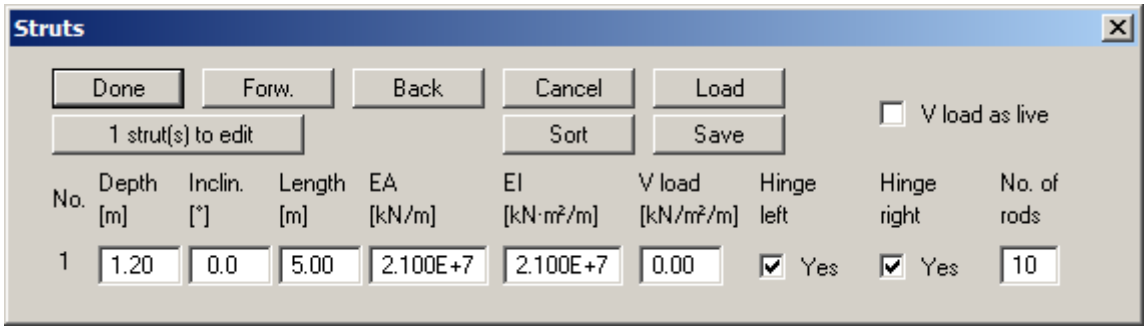
By activating the "**Verification with qs,k**" check box instead of "**H DM**" und "**L GR**" the following data have to be entered:

- FL [m] = free length (top)
- TL [m] = total injection length
- D [m] = diameter

Then, the skin friction values qs,k defined in the menu item "**Editor 1/Soils**" are used for calculation (see Section 8.2.5).

8.3.8 "Struts" menu item

Struts are defined in this dialog box.

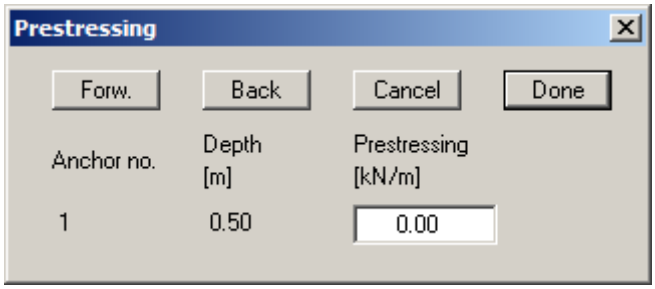


It is necessary to enter the following information:

- Depth [m] = position of the strut head
- Inclination [°] of the strut
- Length [m] of the strut
- EA [kN/m] = axial stiffness of the strut
- EI [kN·m²/m] = bending stiffness of the strut
- V Load [kN/m²/m] = vertical load on the strut
- Hinge (right)
  Here you specify whether the strut is hinged or rigidly fixed to the wall.
- Hinge (left)
  Here you specify the support conditions of the strut on the "opposite side".
  If neither hinge option is selected, the following happens:
  The strut is then modelled with only half length!! On the left-hand edge of the "half" strut complete fixation is assumed. The support can be vertically displaced.
- No. of rods
  If you have defined a vertical load on the strut, you will also get its distribution of moments. Of course, this only works if the strut is divided into a number of rods (finite elements, see Section 7.16). The maximum number of rods that can be specified is 20.

8.3.9 "Prestressing" menu item

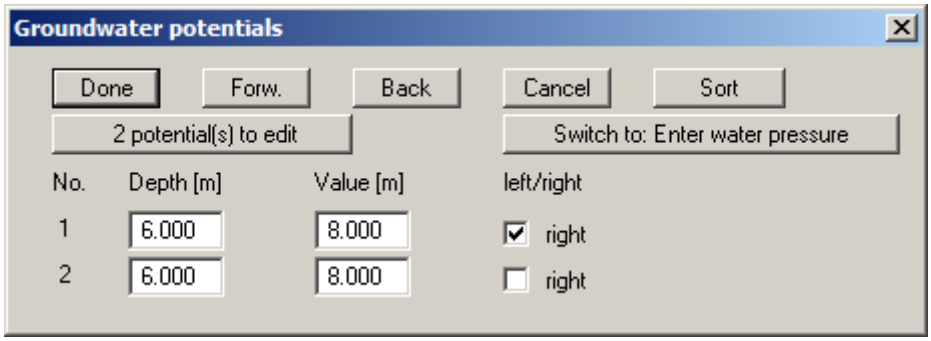
Personal experience has shown that discussions on the necessity of considering prestressing are never-ending. Regardless of these discussions, the program allows prestressing to be taken into consideration.



Positive prestressing must be entered for anchors and struts. Faulty input is pointed out before analysis begins. From a structural perspective, the program inserts a force boundary condition equivalent in size and direction to the prestress into the system at the appropriate depth. Following analysis the prestress is added to the anchor force (strut force). This ensures that deformations only result from forces greater than the prestressing force.

8.3.10 "Groundwater potentials" menu item

You can define additional potentials anywhere along the wall (see Section 7.7.1.2).

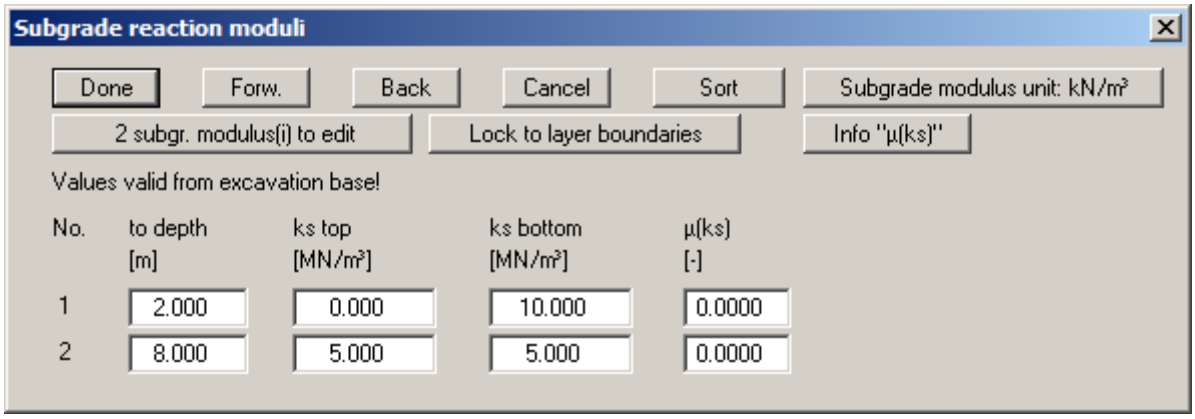


In the above example, potentials of 8.0 m have been entered for 6.0 m below the top of the wall on either side. Additional potentials are only taken into consideration if you selected the water pressure approach using ***flow conduits*** (see Section 7.7.1.2).

Instead of potentials in [m] the data may also be entered as water pressures in [kN/m²] after clicking "**Switch to: Enter water pressure**".

8.3.11 "Subgrade reaction moduli" menu item

If the analysis is to be performed for an elastically bedded wall toe, it is necessary to enter a modulus of subgrade reaction. In this dialog box you can enter the profile of the subgrade reaction moduli.



The number of subgrade reaction moduli can be adapted to your system using the "**x subgr. modulus(i) to edit**" button. It must be noted, however, that in this case the depth data is with reference to the excavation base and are positive downwards, regardless of the initial setting of the "**Use absolute heights**" check box. You can also have the program take care of the depth input by activating the "**Lock to layer boundaries**" check box. Now, you only need to enter the values for the modulus of subgrade reaction.

The following modulus of subgrade reaction profile was modelled using the values shown in the dialog box:

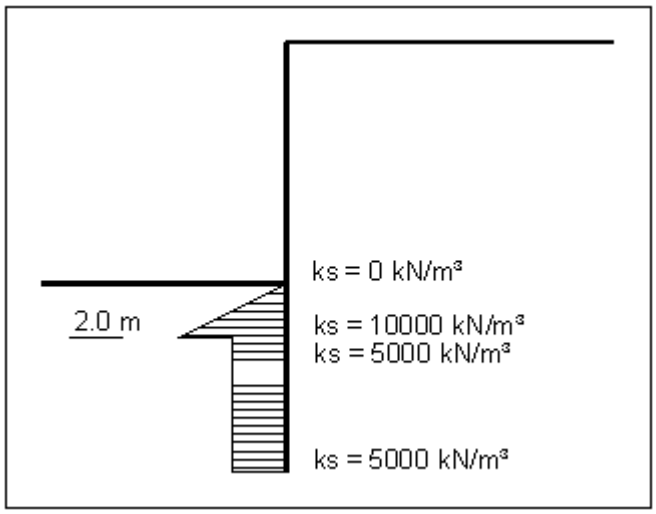
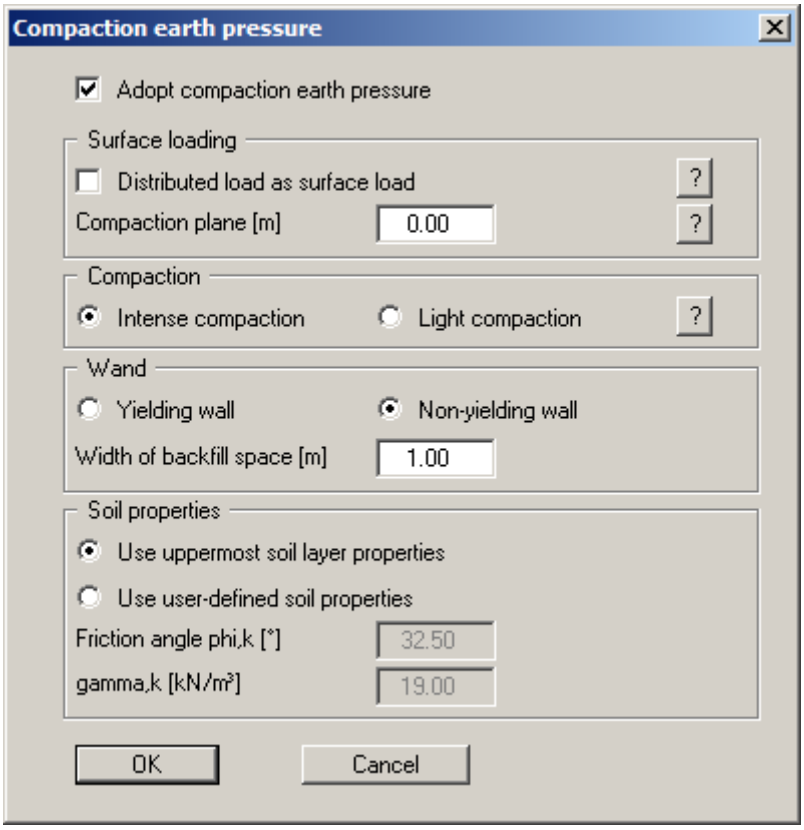


Figure 26 Distribution of subgrade reaction moduli

The tangential bedding is given in the "**μ(ks)**" field as a multiple of the horizontal bedding. However, the factor is generally of little importance, since the wall toe is considered to be vertically non-displaceable, meaning that deformations in a longitudinal direction, and any associated tangential subgrade reaction moduli, are small.

8.3.12 "Compaction earth pressure" menu item

The compaction earth pressure to DIN 4085:2011 can be analysed using this menu item. The data required are described in the DIN and can be entered into the dialog box shown below. Pressing the "**?**" buttons, you will see further information.



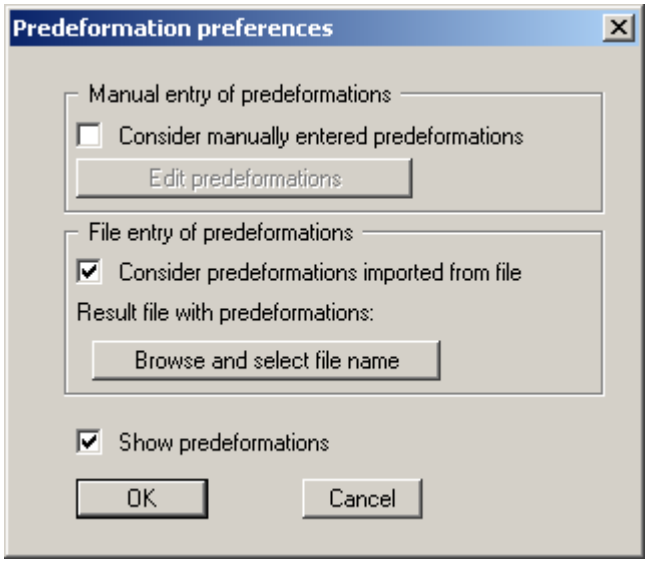
Two files containing analyses taken from *Geotechnische Nachweise nach EC 7 und DIN 1054* (Geotechnical analyses to EC7 and DIN 1054), 3rd edition 2012, by Martin Ziegler, can be found in the program's example folder. The analyses refer to pages 70 and 72 of the literature reference.

8.3.13 "Predeformation information" menu item

You will see information on predeformations.

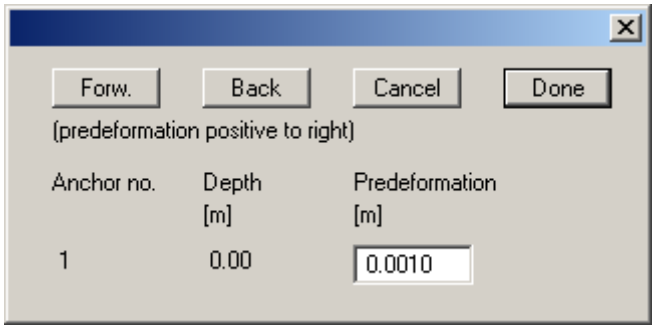
8.3.14 "Predeformation preferences" menu item

If predeformations from previous construction phases must be taken into consideration as boundary conditions (see Section 7.22), the necessary settings are made here.



Predeformations can be defined in two different ways:

- ***Manual entry of predeformations***
  If you know the size of the predeformations, select "**Consider manually entered predeformations**", and then click the "**Edit predeformations**" button.



  In the dialog box that appears you can enter predeformation for every currently defined anchor.

- ***File entry of predeformations***
  Alternatively, predeformation data can be automatically imported from an existing **GGU-RETAIN** file. Select "**Consider predeformations imported from file**" and "**Browse and select file name**". A dialog box will open for selecting the appropriate file, which must have been saved with the results of the analysis; otherwise you will receive an error message. The program imports the appropriate data from this file. The file name is then shown on the button and below this the project identification of the imported file.

  You can view the loaded values by clicking "**Edit predeformations**". If you need to alter the height of the anchors during further processing, these values will be automatically adjusted. This means that whenever you begin an analysis, the selected file will be loaded and the values recomputed. You may also want to recompute the current system in the course of a later session, although the file with the predeformation data has altered in the meantime. In this case, the new predeformation data will be loaded automatically.

In the bottom section of the dialog box you can choose whether or not to have the loaded predeformation data shown on screen. If you choose to have it shown, the system graphics show the system's bending line with predeformations.

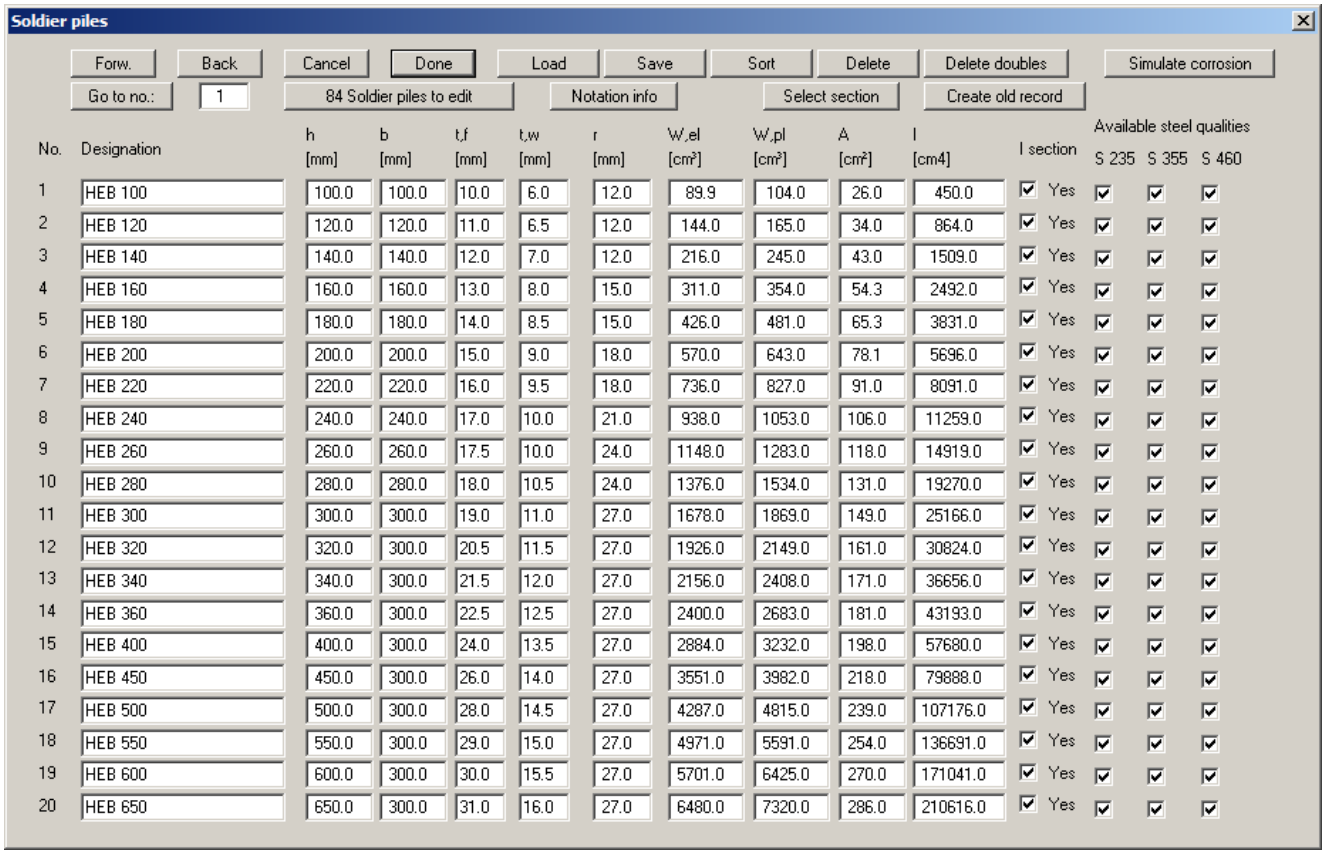
8.3.15 "Soldier pile"/"Sections"/"Bored pile wall"/"Diaphragm wall"/ "Contiguous pile wall"/"Girder"/"Section data" menu items

8.3.15.1 General note

This menu item contains various designations and dialog boxes, depending on the type of wall selected in "**File/New**" or "**Editor 1/Analysis options**" and on whether the section values are from the section list or user-defined.

8.3.15.2 "Soldier piles" menu item

If you selected the "**Soldier pile wall**" and "**Using section list**" check boxes in the "**File/New**" or "**Editor 1/Analysis options**" menu items, you will see the following dialog box with the currently loaded list of soldier piles. For this view the "**Steel design to EC 3**" check box in the "**Editor 1/Analysis options**" menu item must be activated.

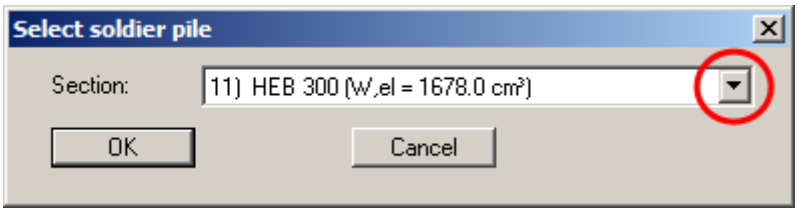


No.	Designation	h [mm]	b [mm]	t,f [mm]	t,w [mm]	r [mm]	W,el [cm³]	W,pl [cm³]	A [cm²]	I [cm4]	I section	Available steel qualities S 235	S 355	S 460
1	HEB 100	100.0	100.0	10.0	6.0	12.0	89.9	104.0	26.0	450.0	Yes	☑	☑	☑
2	HEB 120	120.0	120.0	11.0	6.5	12.0	144.0	165.0	34.0	864.0	Yes	☑	☑	☑
3	HEB 140	140.0	140.0	12.0	7.0	12.0	216.0	245.0	43.0	1509.0	Yes	☑	☑	☑
4	HEB 160	160.0	160.0	13.0	8.0	15.0	311.0	354.0	54.3	2492.0	Yes	☑	☑	☑
5	HEB 180	180.0	180.0	14.0	8.5	15.0	426.0	481.0	65.3	3831.0	Yes	☑	☑	☑
6	HEB 200	200.0	200.0	15.0	9.0	18.0	570.0	643.0	78.1	5696.0	Yes	☑	☑	☑
7	HEB 220	220.0	220.0	16.0	9.5	18.0	736.0	827.0	91.0	8091.0	Yes	☑	☑	☑
8	HEB 240	240.0	240.0	17.0	10.0	21.0	938.0	1053.0	106.0	11259.0	Yes	☑	☑	☑
9	HEB 260	260.0	260.0	17.5	10.0	24.0	1148.0	1283.0	118.0	14919.0	Yes	☑	☑	☑
10	HEB 280	280.0	280.0	18.0	10.5	24.0	1376.0	1534.0	131.0	19270.0	Yes	☑	☑	☑
11	HEB 300	300.0	300.0	19.0	11.0	27.0	1678.0	1869.0	149.0	25166.0	Yes	☑	☑	☑
12	HEB 320	320.0	300.0	20.5	11.5	27.0	1926.0	2149.0	161.0	30824.0	Yes	☑	☑	☑
13	HEB 340	340.0	300.0	21.5	12.0	27.0	2156.0	2408.0	171.0	36656.0	Yes	☑	☑	☑
14	HEB 360	360.0	300.0	22.5	12.5	27.0	2400.0	2683.0	181.0	43193.0	Yes	☑	☑	☑
15	HEB 400	400.0	300.0	24.0	13.5	27.0	2884.0	3232.0	198.0	57680.0	Yes	☑	☑	☑
16	HEB 450	450.0	300.0	26.0	14.0	27.0	3551.0	3982.0	218.0	79888.0	Yes	☑	☑	☑
17	HEB 500	500.0	300.0	28.0	14.5	27.0	4287.0	4815.0	239.0	107176.0	Yes	☑	☑	☑
18	HEB 550	550.0	300.0	29.0	15.0	27.0	4971.0	5591.0	254.0	136691.0	Yes	☑	☑	☑
19	HEB 600	600.0	300.0	30.0	15.5	27.0	5701.0	6425.0	270.0	171041.0	Yes	☑	☑	☑
20	HEB 650	650.0	300.0	31.0	16.0	27.0	6480.0	7320.0	286.0	210616.0	Yes	☑	☑	☑

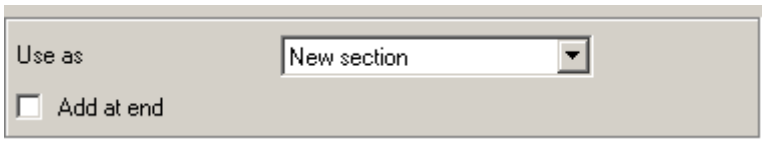
The following options are available:

- You can navigate through the list using "**Forw.**" and "**Back**". "**Go to no.**" allows you to jump to the soldier pile specified.
- "**Cancel**", "**Done**"
  Exit the dialog box either saving or rejecting your modifications using these buttons.
- "**x Soldier piles to edit**", "**Notation info**"
  Using this button you may expand or reduce the list of soldier piles. New soldier piles are added to the end of the list. A description of the abbreviations used to enter the section data is available via the "**Notation info**" button.

- "**Load**", "**Save**", "**Sort**", "**Delete**", "**Delete doubles**"
  A different soldier pile list can be opened by pressing the "**Load**" button. It is then possible to append the new list to an already open soldier pile list. After appending soldier piles it may be expedient to delete any double sections in the list by pressing the "**Delete doubles**" button. The soldier piles can then be sorted either by moment of inertia or by name by pressing the "**Sort**" button. The modified soldier pile list may then be saved to the program folder as "**.tbw_ggu**" file for subsequent analyses by pressing the "**Save**" button. Any number of sections in a sequence in the list can be deleted using the "**Delete**" button.
- "**Select section**"
  A dialog box opens in which you select the required soldier pile for design.



- "**Simulate corrosion**"
  You can define the corrosion for any soldier pile, which is then used to calculate the new section data. The new section data can be allocated to the selected soldier pile, or to a different, existing soldier pile. The data can also be allocated to a new soldier pile, which is added to the existing list (also see Worked example 2, Section 6.2.2).



- "**Create old record**"
  Using this button you can save a record containing the old section tables based on the global safety factors ("**.tbw**", "**.spw**").

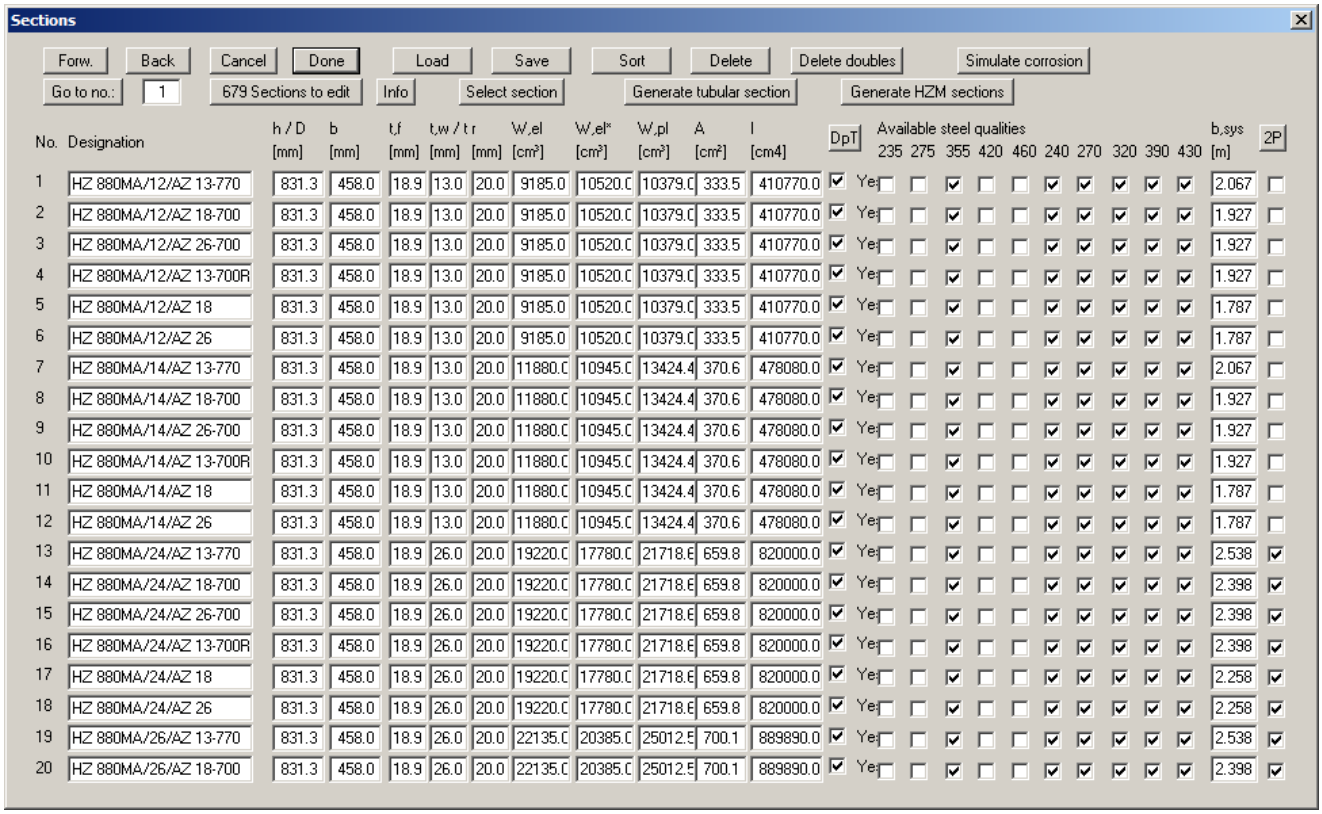
If ***Steel design to EC 3*** is not activated, the old dialog box opens. A check box in front of the number of each soldier pile can be activated. This is used to select the respective soldier pile. If you then press the "**Selected section as design section**" button, this section is used as the design section. The section parameters ("**h**" = height; "**b**" = width; "**A**" = area; "**I**" = moment of inertia) can be edited. The program uses **I** and **h** to determine the section modulus **W** during the subsequent design phase. The variable **S** is the first moment of area of the section and **s** is the web thickness. The value of "**S/s**" is required for verification of shear stress.

8.3.15.3 "Sections" menu item

Upon selecting "**Sheet pile wall**" and "**Using section list**" you see a dialog box which appears almost exactly the same as that for soldier piles. Here, too, the "**Steel design to EC 3**" check box must be activated. The sheet pile wall sections can be stored in a "**.spw_ggu**' file.

If you do not design to EC 3, it is important to note that for sheet pile wall sections the area "**A**" must be entered in [cm²/m] and the moment of inertia "**I**" in [cm^4/m]. Also note that the section values given in the table assume that a shear resistant connection exists between the supplied double sections. The variable **S** is the first moment of area of the section and **s** is the web thickness. The "**alp**" variable is the opening angle of the section. The value of "**S·sin(alp)/s**" is required for verification of shear stress.

If "**Combined sheet pile wall**" is selected, the sections dialog box is expanded slightly:



No.	Designation	h / D [mm]	b [mm]	t,f [mm]	t,w / t r [mm]	[mm]	W,el [cm³]	W,el* [cm³]	W,pl [cm³]	A [cm²]	I [cm4]	DpT	Available steel qualities 235	275	355	420	460	240	270	320	390	430	b,sys [m]	2P
1	HZ 880MA/12/AZ 13-770	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.067	☐
2	HZ 880MA/12/AZ 18-700	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
3	HZ 880MA/12/AZ 26-700	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
4	HZ 880MA/12/AZ 13-700R	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
5	HZ 880MA/12/AZ 18	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.787	☐
6	HZ 880MA/12/AZ 26	831.3	458.0	18.9	13.0	20.0	9185.0	10520.0	10379.0	333.5	410770.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.787	☐
7	HZ 880MA/14/AZ 13-770	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.067	☐
8	HZ 880MA/14/AZ 18-700	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
9	HZ 880MA/14/AZ 26-700	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
10	HZ 880MA/14/AZ 13-700R	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.927	☐
11	HZ 880MA/14/AZ 18	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.787	☐
12	HZ 880MA/14/AZ 26	831.3	458.0	18.9	13.0	20.0	11880.0	10945.0	13424.4	370.6	478080.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	1.787	☐
13	HZ 880MA/24/AZ 13-770	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.538	☑
14	HZ 880MA/24/AZ 18-700	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.398	☑
15	HZ 880MA/24/AZ 26-700	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.398	☑
16	HZ 880MA/24/AZ 13-700R	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.398	☑
17	HZ 880MA/24/AZ 18	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.258	☑
18	HZ 880MA/24/AZ 26	831.3	458.0	18.9	26.0	20.0	19220.0	17780.0	21718.6	659.8	820000.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.258	☑
19	HZ 880MA/26/AZ 13-770	831.3	458.0	18.9	26.0	20.0	22135.0	20385.0	25012.5	700.1	889890.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.538	☑
20	HZ 880MA/26/AZ 18-700	831.3	458.0	18.9	26.0	20.0	22135.0	20385.0	25012.5	700.1	889890.0	☑	Yes	☐	☐	☑	☐	☐	☑	☑	☑	☑	☑	2.398	☑

You can have new sections generated by the program using the "**Generate tubular section**" button. The new sections created here can be saved in a "**.kom_ggu**" file.

8.3.15.4 "Bored pile wall"/"Diaphragm wall"/"Contiguous wall" menu items

If you specified "**Bored pile wall**" or "**Contiguous wall**" and "**Using section list**", a dialog box opens for entering the "**Bored pile diameter**".

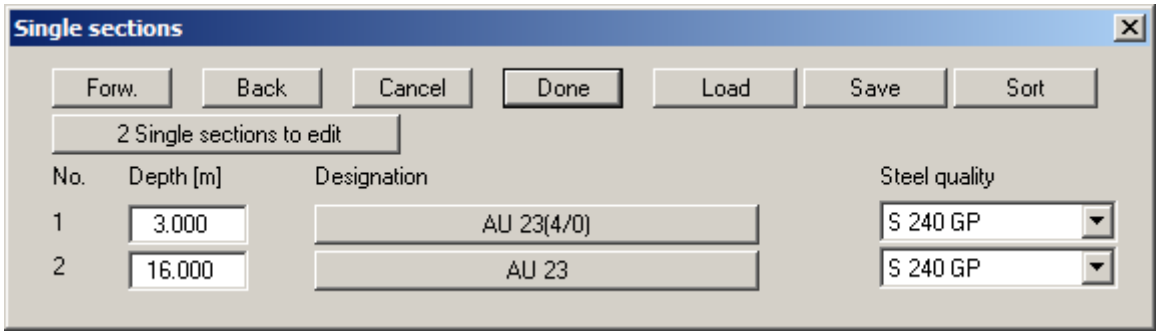
If you select "**Diaphragm wall**" and "**Using section list**" you must enter the "**Diaphragm wall thickness**" in a dialog box.

8.3.15.5 "Girder" menu item

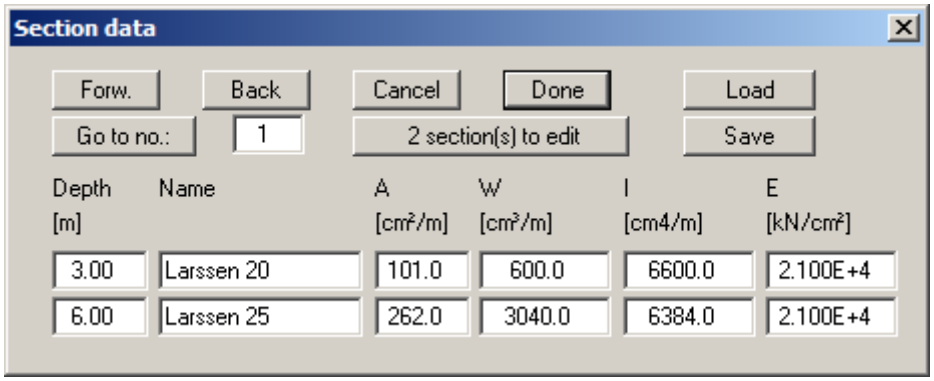
If you selected the "**CMG wall**" and "**Using section list**" check boxes in the "**File/New**" or "**Editor 1/Analysis options**" menu items, you will see the following dialog box with the currently loaded list of soldier piles (see Section 8.3.15.2).

8.3.15.6 "Section data" menu item

If you activated the "**Using user-defined section data**" check box in the "**File/New**" or "**Editor 1/Analysis options**" menu items, you can define several sections for the wall here. This allows partial corrosion of a sheet pile wall to be modelled, for example (see Worked example 2, Section 6.2.3).



If ***Steel design to EC 3*** is not activated, the following dialog box opens:



The "**Depth**" from the top of the wall or the absolute heights (of the section) have been entered, the section "**Name**", the area "**A**", the section modulus "**W**", the moment of inertia "**I**" and Young's modulus "**E**".

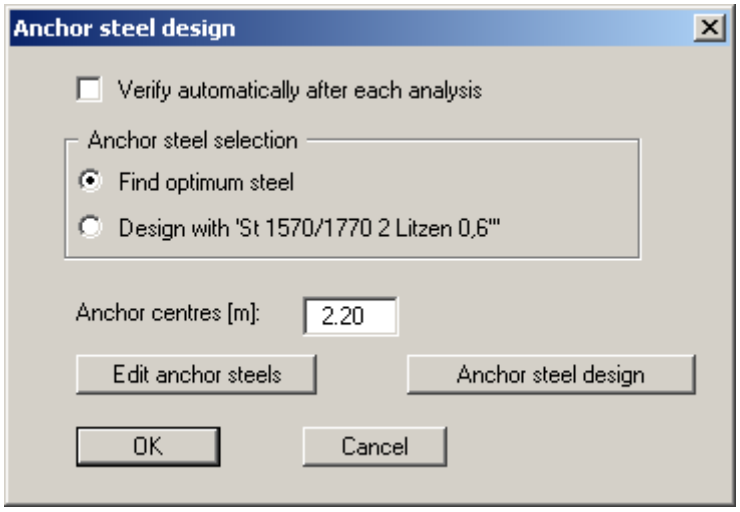
8.3.16 " Young's modulus/Specific weight"/"Specific weight" menu items

This menu item opens two different dialog boxes depending on whether you are working with a section list or with user-defined section data:

- "**E modulus/Specific weight**"
  When working with the ***section list***, you must enter Young's modulus and specific weight of the wall. From the former the bending line is calculated, from the latter the self-weight of the wall.
- "**Specific weight**"
  Using ***user-defined section data*** Young's modulus is entered in the corresponding dialog box, so that here it is only necessary to specify the specific weight of the wall.

8.3.17 "Anchor steel design" menu item

Using this menu item you can specify preferences for designing the anchor steel and carry out the design for an analysed system by means of the "**Anchor steel design**" button. If you activate the "**Verify automatically after each analysis**" check box, you will be automatically presented with a query for anchor steel design when analysis is complete.



You can choose to design with a specified anchor steel or to search for the optimum steel from a list of existing steels.

For an analysis using ***global safety factors*** you must then enter the factor for the anchor steels which for active earth pressure = 0.0 and for loading solely by at-rest pressure = 1.0. Intermediate values can be linearly interpolated. FOS equals 1.0 for prestressed anchors.

When using ***partial safety factors*** only the anchor centres still need to be defined, as the other data is taken into consideration via the partial factor for permanent actions resulting from at-rest earth pressure.

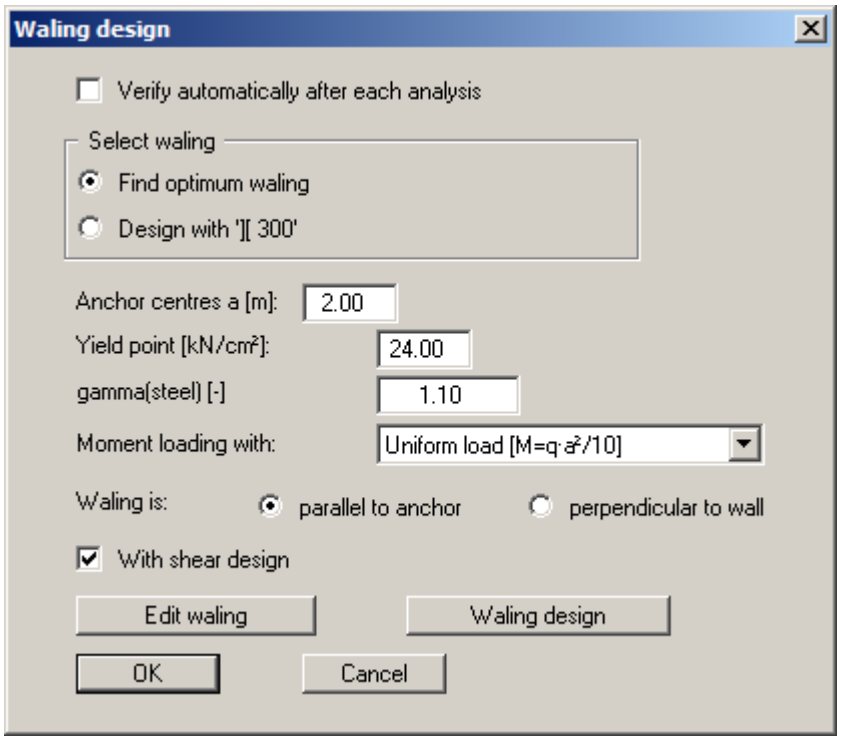
Using the "**Edit anchor steels**" button you arrive at a dialog box listing the existing anchor steels with their names and the allowable anchor forces for active and at-rest earth pressures. By marking the selection check box in front of the desired anchor steel and clicking the "**Selected steel as design steel**" button, it can be transferred to the above dialog box. New anchor steels can be added to the list after clicking "**x anchor steel(s) to edit**".

> The "**Anchor steel design**" button only appears if the system is already analysed!

8.3.18 "Waling design" menu item

Using this menu item you can specify preferences and then carry out the design for an analysed system by means of the "**Waling design**" button. The bending and shear verifications of the waling are then displayed on the screen.

If you activate the "**Verify automatically after each analysis**" check box, you will be automatically presented with a query for waling design when analysis is complete.



You can choose to design with a specified waling or to search for the optimum waling from a list of existing waling. When using ***partial factors*** you must enter the "**Yield point**" and "**gamma(steel)**", and the allowable stresses for the ***global factors***. Moreover, you must also enter the "**Anchor centres**" and select the type of "**Moment loading**" on which the design is based. Finally, you can specify whether the waling runs parallel to the anchor or perpendicular to the wall.

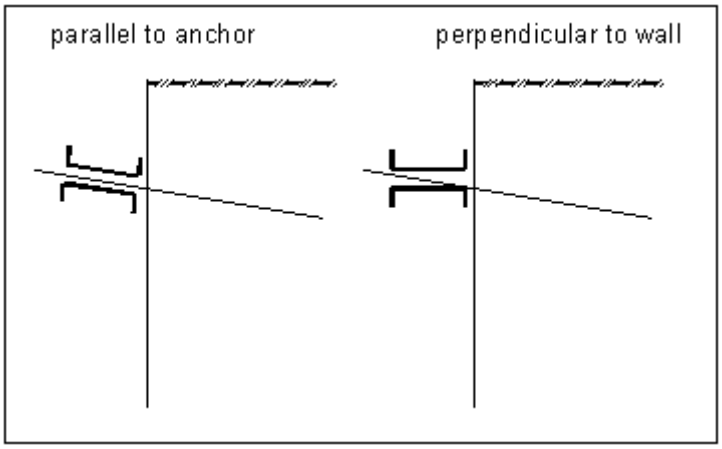


Figure 27 Walings

The "**Edit waling**" button opens a dialog box with a list of walings showing their names, section moduli and the values of "**S/s**" (first moment of area and web thickness) required for shear design. Here, you can select a waling and use it as the design waling using the button "**Selected waling as design waling**". New waling can be entered by clicking "**x waling(s) to edit**" and specifying a new number.

8.4.1 "Info" menu item

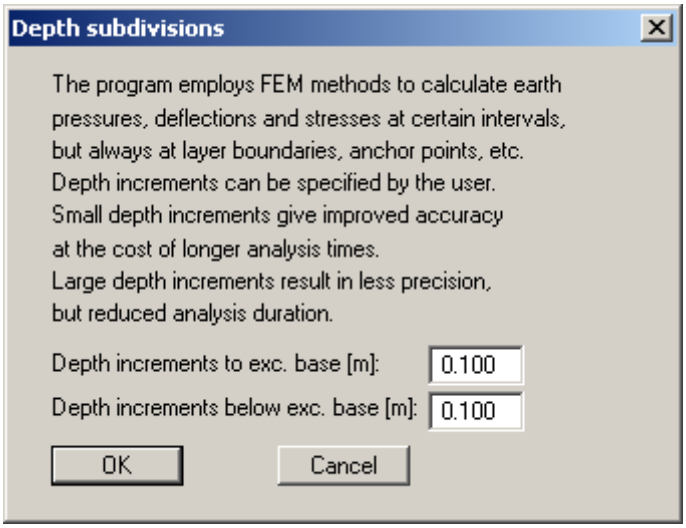
You will see information on the current system in a message box.

8.4.2 "Special preferences" menu item

The program performs a multitude of plausibility checks s. After starting the analysis the preferences specified by the user are displayed in a message box; for problematical preferences separate information or warning are displayed. It is therefore recommended to leave the "**Show warnings in future**" check box activated. If you do not want to see the automatic display when the analysis starts, deactivate the check box. You can subsequently view your special preferences using this menu item.

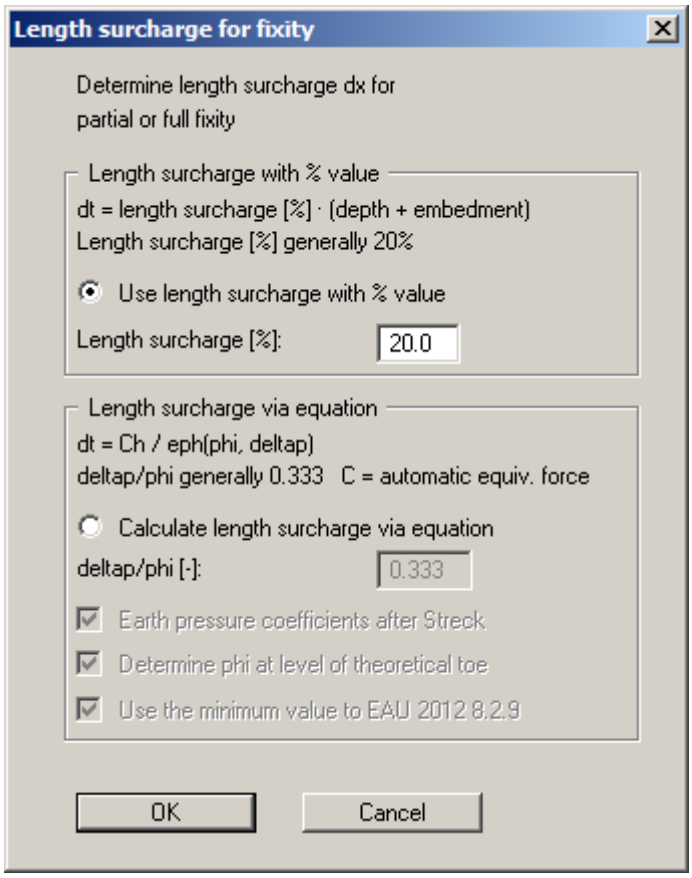
8.4.3 "Depth subdivisions" menu item

GGU-RETAIN uses the finite element method, which requires the system to be divided into a number of finite elements (rods) (see Section 7.16). You can specify the size of these depth increments for the region above and below the excavation base.



8.4.4 "Length surcharge" menu item

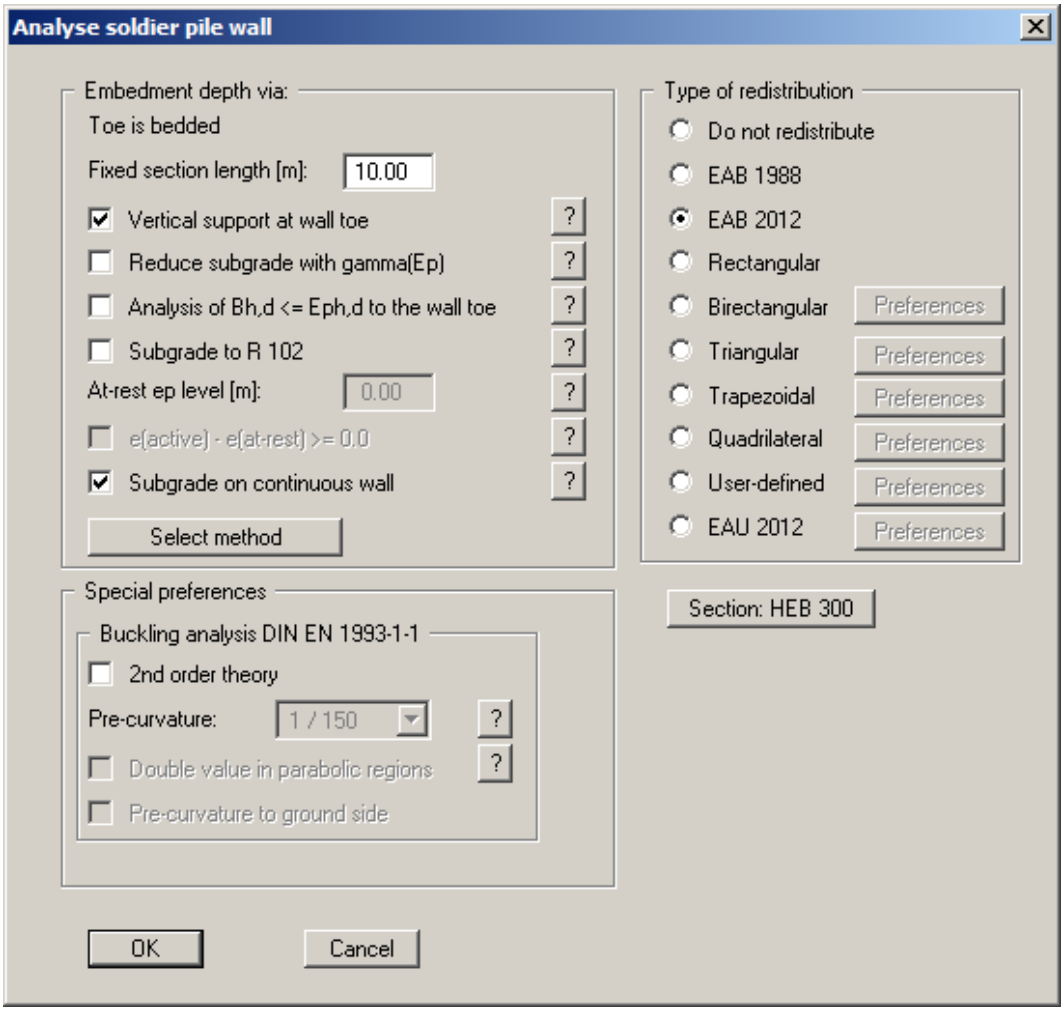
Here you can specify the way in which length surcharge is calculated for full or partial wall fixity (see Section 7.15).



8.4.5 "Analyse" menu item

8.4.5.1 Start dialog box

Once you have entered all data required to fully describe the system it can be analysed. After going to the "**System/Analyse**" menu item a ***start dialog box***, divided into three group boxes, opens (see Sections 8.4.5.2 to 8.4.5.4 for descriptions). You can also initiate the analysis using the [**F5**] function key and see the same start dialog box. The available options and check boxes vary depending on the analysis principles adopted. At the bottom right of the dialog box a button can be seen labelled with the currently selected design section. If you select this button you can view and edit the current values for the design section.



When all the options are selected as you require, you can start the analysis by clicking "**OK**". Initially, **GGU-RETAIN** carries out a comprehensive plausibility check of the data you have entered and informs you of any inconsistencies. The system is then analysed. Information relating to the current stage of the analysis is displayed in the title bar. The analysis process can be terminated at any time (perhaps because you want to alter one of the parameters) by clicking the right mouse button.

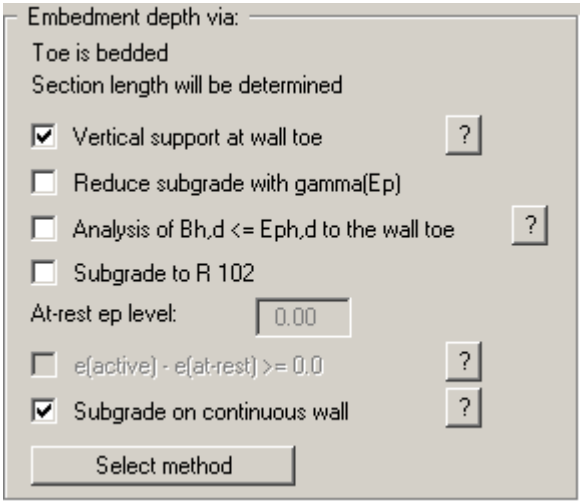
Once the analysis is complete the design can follow by clicking "**Yes**" in the prompt. The design phase is more closely described in Section 8.4.6 ("**System/Design defaults**"). If "**No**" is now clicked or following completion of design, the results are presented in message boxes and then visualised graphically on the screen. The system data can be saved in a file, together with the results (menu item "**File/Save as**", Section 8.1.4).

8.4.5.2 "Embedment depth via:" group box

In the "**Embedment depth via**" group box of the start dialog box of the "**System/Analyse**" menu item you can define the method of calculation for the embedment depth.

The default is "**Toe is free or fixed**", the section length is determined automatically. The degree of fixity is entered below this. If you prefer to use a different method to determine the embedment depth, use the "**Select method**" button. Select the method from the four options available. If you select one of the two options using fixed section lengths "**Fixed section length**" appears in the start dialog box instead of the degree of fixity; you must enter the value you require.

Where a ***bedded toe*** is used, the subgrade reaction can be reduced using the partial factor gamma(Ep) by activating the check box. The reduction can be made using characteristic values to DIN 1054, GL. (47) and (48). If the "**Reduce subgrade with gamma(Ep)**" check box is not activated, the subgrade reaction can be taken into consideration in accordance with the EAB, R 102 ("Modulus of subgrade reaction method", published by Prof. Anton Weißenbach in *Bautechnik* (Construction Engineering) 80/2003, Issue 2).



In the subgrade zone the difference between the active earth pressure and the at-rest earth pressure is adopted as a load acting on the wall. If the "**e(active) -e(at-rest) >= 0.0**" check box is activated this difference cannot become smaller than "**0.0**".

If the "**Subgrade on continuous wall**" check box is deactivated, the subgrade will be adopted for the girder width.

8.4.5.3 "Special preferences" group box

In the "**Special preferences**" group box of the start dialog box of the "**System/Analyse**" menu item you can specify whether any ***area loads*** should be included in an earth pressure redistribution. You may "**Redistribute ep to toe**" and not, as is usual, to the load transition point.

For an analysis employing the ***global safety factors***), the load transition point can be calculated with or without any prevalent water pressure ("**Transition point with wp**" check box). In addition, when analysing with the old safety factors you can specify whether the program should place the passive earth pressure in front of the wall. If the "**Passive ep in front**" check box is not activated, the passive and active earth pressures are added (superimposed). The new standard does not allow addition of the active earth pressure (action) and the passive earth pressure (resistance). The "**Passive ep in front**" check box therefore does not appear if the ***partial safety factor concept*** is selected, because the passive earth pressure is always in front.

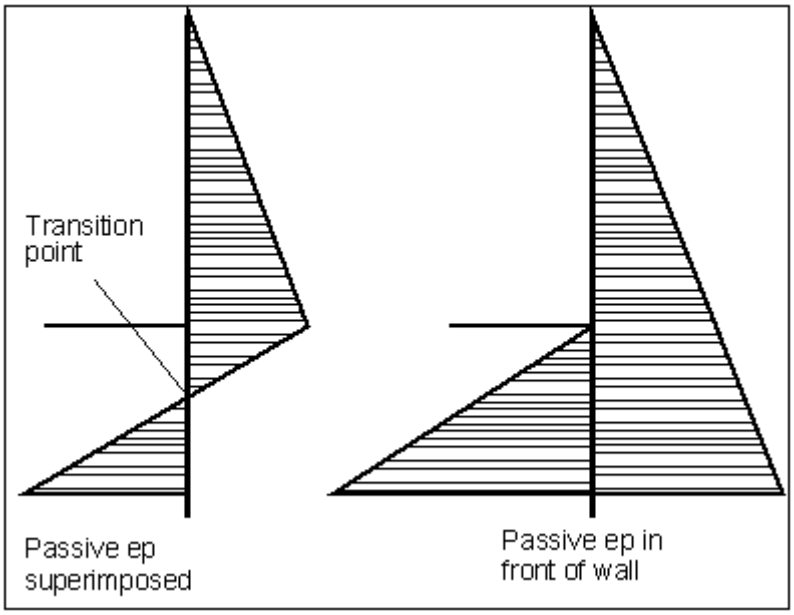


Figure 28 Passive earth pressure (ep) (in front or superimposed)

With a bedded wall toe and a least two rows of anchors the toe area is subjected to the total active earth pressure. Passive earth pressure, which is greater, plays no part in the system's statics, but is used in comparing soil pressures with passive earth pressure, so as to be able to reduce embedding if necessary. Thus, statically the wall toe acts like a heavily loaded cantilever arm with a correspondingly large moment in the area of the lower anchor. Particularly with soft bedding and very stiff sections, compared with a calculation with superimposed passive earth pressure, an excessively large moment can be the result. **GGU-RETAIN** will alert you to such situations. Which of the two approaches is the right one, is up to the user to decide.

Retaining walls subject to buckling hazards can be analysed using "**2nd order theory**". A system pre-curvature, which can be adopted at 1/150, must be defined for buckling analysis to DIN EN 1993-5 or DIN EN 1993-1-1. The direction of pre-curvature must also be defined (to the ground side or the atmosphere side). Whether a pre-curvature towards the ground side or the atmosphere side provides the more unfavourable design values is system-dependent: it is therefore absolutely vital that both directions are investigated in a buckling analysis (see "**Theoretical principles/2nd order theory**", Section 7.18).

8.4.5.4 "Type of redistribution" group box

In the "**Type of redistribution**" group box of the start dialog box of the "**System/Analyse**" menu item the following options are available:

- "**Do not redistribute**"
  The analysis is performed using classical earth pressure redistribution.
- "**EAB 1988**" and "**EAB 2012**"
  For soldier pile walls and sheet pile walls (in-situ concrete walls), the EAB 1988 and EAB 2012 provide redistribution figures, depending on the location of anchors. **GGU-RETAIN** will automatically choose the appropriate figure. If no suitable figure is found, an error message will appear.
- "**Rectangular**"
  Earth pressure is redistributed in the shape of a rectangle.
- "**Birectangular**"
  Earth pressure is redistributed in birectangular shape. The relationship between the top and bottom earth pressure ordinates (eaho/eahu), as well as depth of the subdivision x, can be specified.

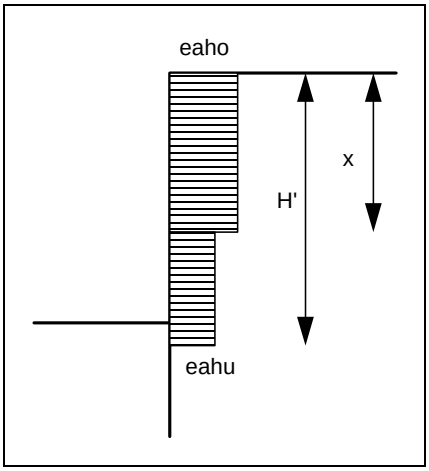


Figure 29 Birectangular earth pressure redistribution

- "**Triangular**"
  Earth pressure is redistributed in the shape of a triangle. The associated "**Preferences**" button enables you to determine the position of the maximum (top, central, bottom).

- "**Trapezoidal**"
  Earth pressure is redistributed in the shape of a trapezoidal. The associated "**Preferences**" button enables you to determine the eahu/eaho ratio.

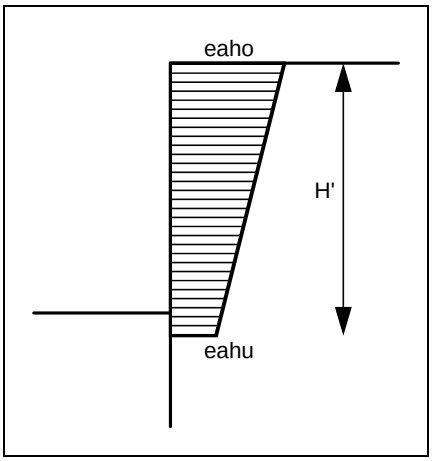


Figure 30 Earth pressure redistribution in a trapezoidal

- "**Quadrilateral**"
  The earth pressure is redistributed in a quadrilateral. After clicking the "**Preferences**" button you can select the ordinates at which the maximum should occur, either by entering the depth or, alternatively, the anchor positions. Activate the appropriate check boxes at the left of the dialog box. The ordinate at the load transition point is defined by the ratio eaho/eahu.

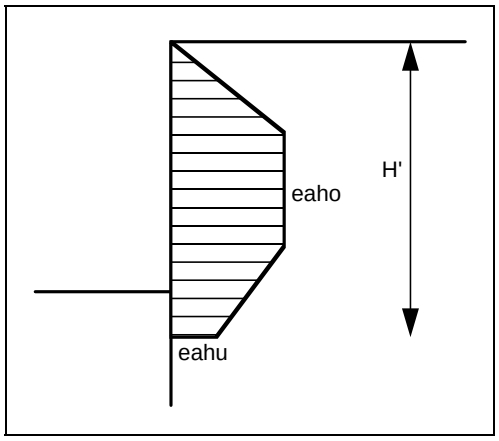
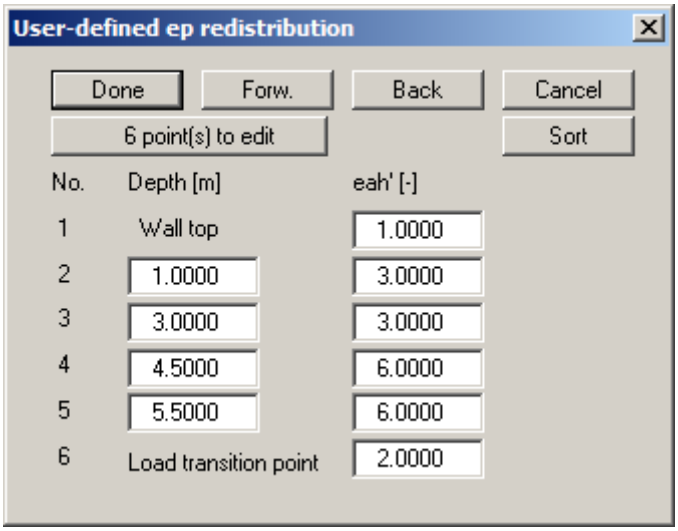


Figure 31 Earth pressure redistribution in a quadrilateral

- **"User-defined"**
  If none of the offered redistribution figures meet your requirements, you have the option of creating your own by defining a polygon.



You can define a number of depths between the top of the wall and the transition point, to each of which you can appoint appropriate earth pressure ordinates. In subsequent computations, earth pressure will be redistributed in exactly the area defined by the polygon you have created. Using the example in the above dialog box the following diagram is obtained:

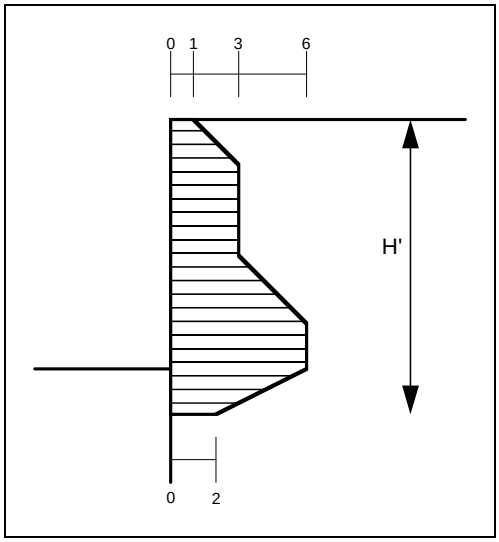


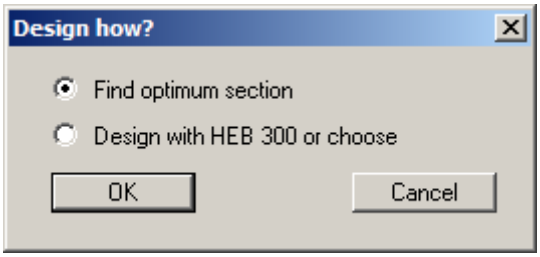
Figure 32 User-defined earth pressure redistribution

- **"EAU 2012"**
  The earth pressure is redistributed in accordance with EAU 2012. After clicking on the "**Preferences**" button you can differentiate between an excavated or a backfilled wall.

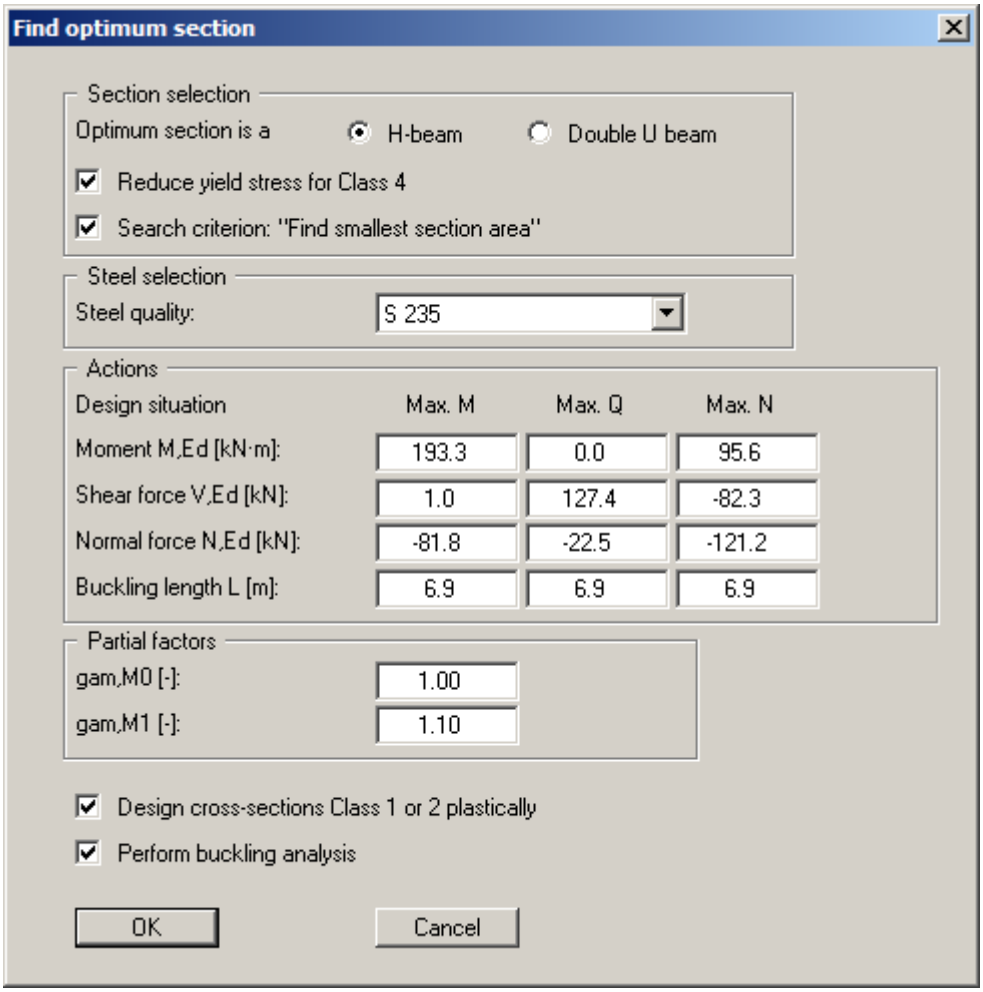
8.4.6 "Design defaults" menu item

Once the current system has been analysed you can progress directly to the retaining wall design (see menu item "**System/Analyse**", Section 8.4.5.1). In addition, using this menu item, you can also carry out subsequent design for a different section to that used for the analysis.

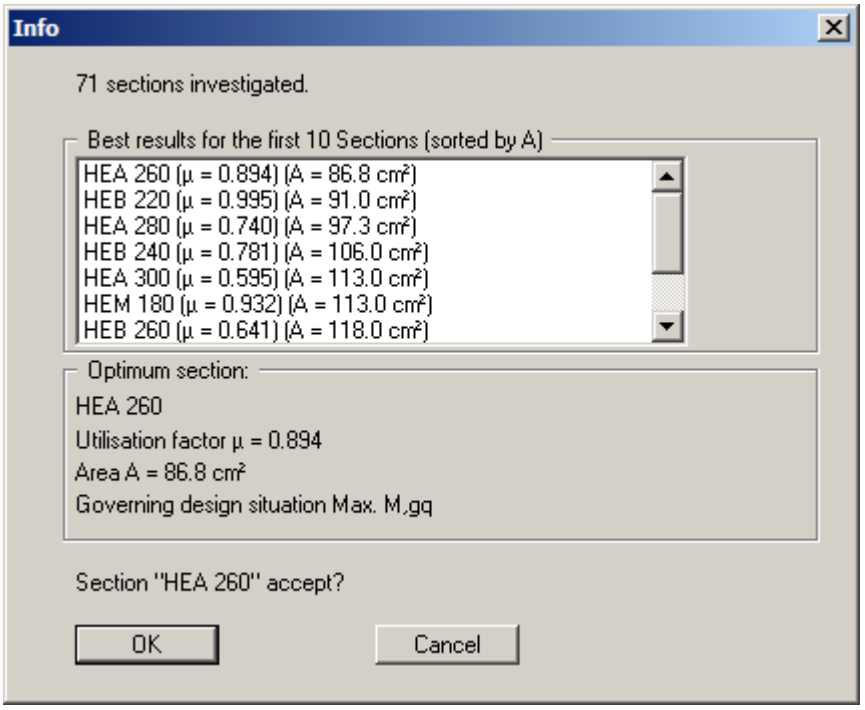
The program reacts differently depending on the type of retaining wall (soldier pile wall, sheet pile wall or in-situ concrete wall). For ***soldier pile walls***, for example, you will see the following dialog box:



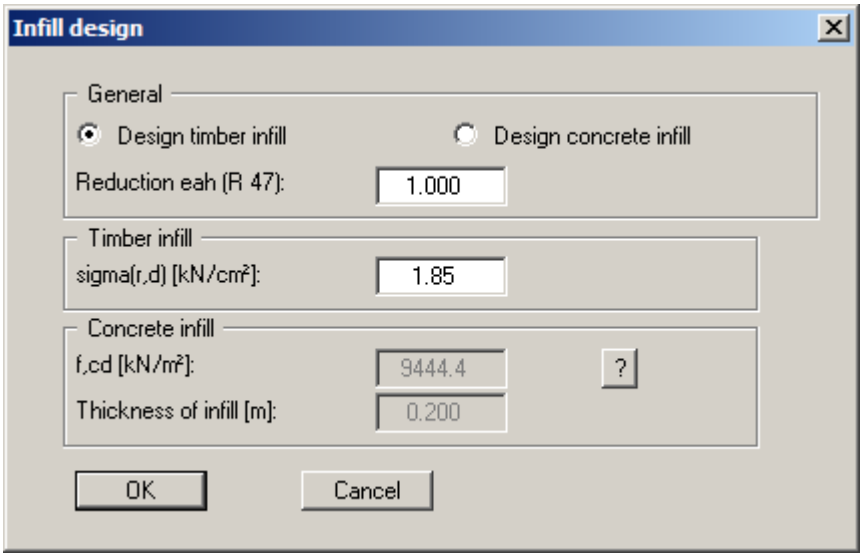
If you would prefer to have the program search for the optimum section, select the upper radio button and the following dialog box opens:



The search result is displayed in another dialog box. If you want to adopt the optimum section proposed by the program, click "**OK**".



Following this the analysis results are displayed in a message box. If you selected the proposed optimum soldier pile, you should adapt the soldier pile width to suit the section in your system by pressing the "**Accept**" button. Subsequent reanalysis is recommended. The infill walling for soldier pile walls can then be designed.



The program then searches for the maximum active earth pressure ordinate, *max eah*, determines the maximum moment for the infill from

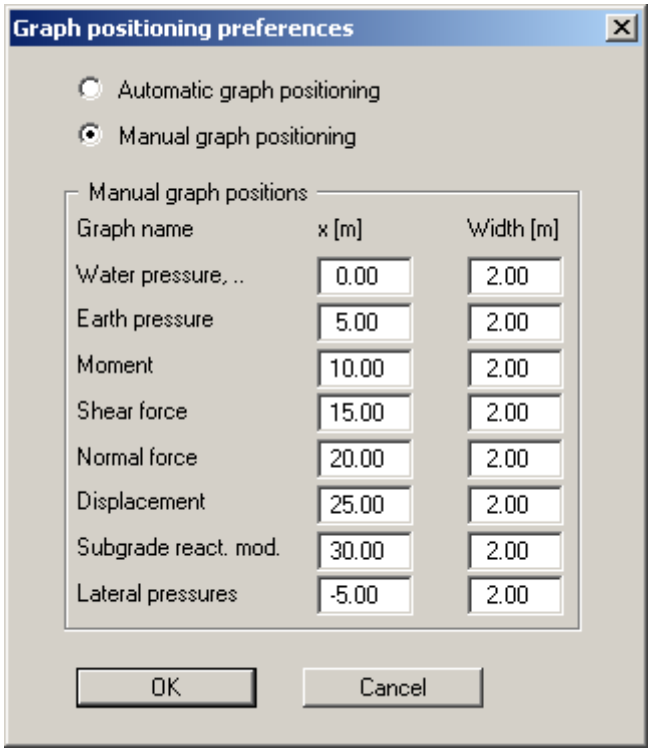
(max eah) · (soldier pile centres)^2 / 8

and calculates the required thickness of the infill elements using the allowable stress. Alternatively to timber infill, concrete infill can also be designed.

When design is complete you are informed of the verification results in an information box and are presented with the principal data on which the design was based. After confirming with "**OK**", the state variables and the system are presented on the screen.

8.4.7 "Graph positioning preferences" menu item

If you are not happy with the automatic graph arrangement you can arrange them to suit your needs using this menu item. First, activate the "**Manual graph positioning**" radio button.

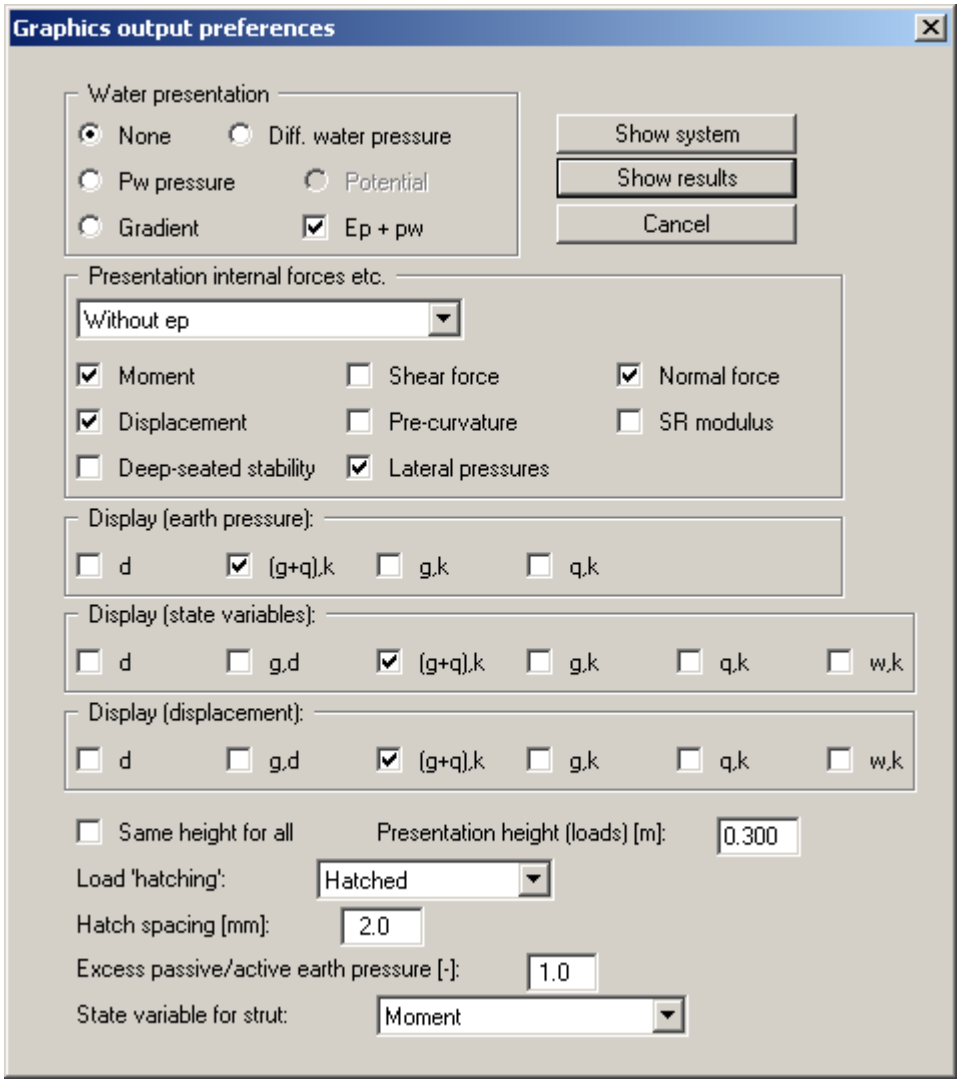


The graphs will then be shown central at the position "**x**" with the specified "**Width**".

> The fastest way to modify the position of a graph is to press the [**F11**] function key and then to pull the graph to the new position holding the left mouse button pressed.

8.4.8 "Graphics output preferences" menu item

Among other things, the screen graphics consist of several graphs, presenting depth-oriented results. This menu item opens a dialog box which allows you to visualise the selected state variables on the screen by activating the corresponding check boxes.



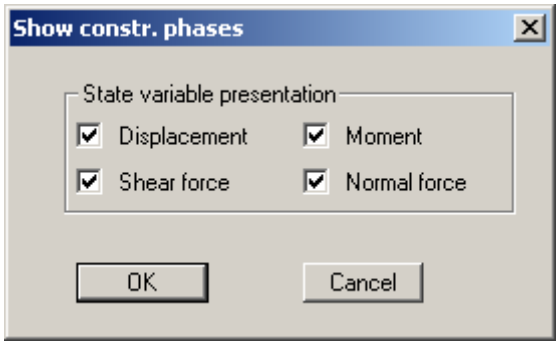
On the whole, the dialog box is self-explanatory. When "**ep + pw**" is selected, the sum of earth pressure and water pressure (pw) is displayed in the diagram for earth pressure. If a system has been analysed using 2nd order theory with buckling analysis to DIN EN 1993-1-1 activated, either the "**Displacement**" or the "**Pre-curvature**" can be selected for display in the result graphics in the dialog box shown above. Any adopted pre-curvature will always be given in the output table.

When analysing using ***partial safety factors*** you will also see the group boxes "**Display (earth pressure):**", "**Display (state variables):**" and "**Display (displacement):**", in which you can activate display of the permanent (g) and/or live loads (q). In addition, the design values (d) can be displayed.

You can also specify hatching and the presentation height of loads. If the "**Same height for all**" check box is not selected, load visualisation is based on load size, the height of the presentation indicating the maximum load.

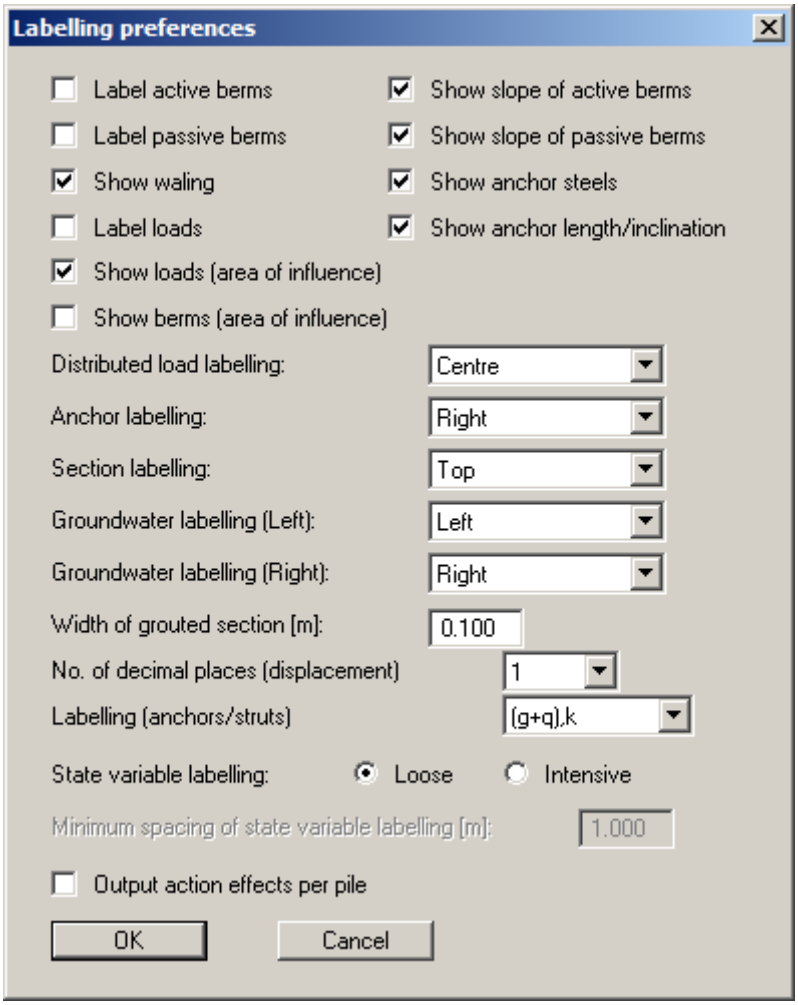
Leave the dialog box by pressing "**Show system**". If the system has already been analysed you can leave the box by pressing "**Show results**" and then view the result graphics on the screen.

When you are working with a particular construction phase the following dialog box appears:



8.4.9 "Labelling preferences" menu item

This menu item allows you to specify labelling preferences for the system visualisation and the result graphics.



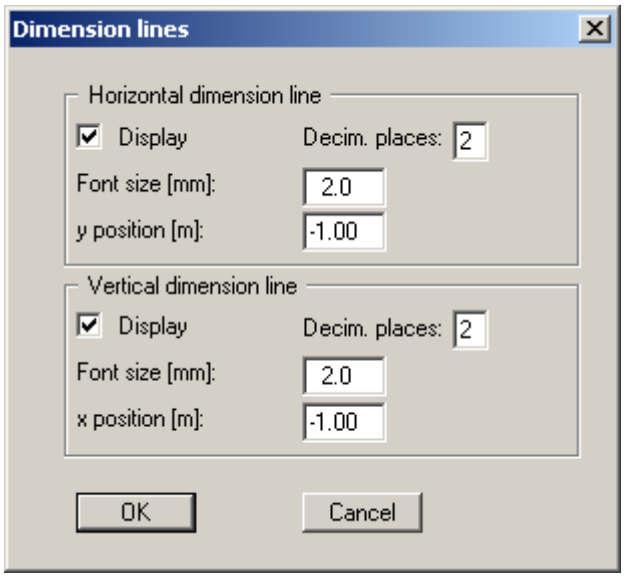
In the dialog box, you activate the required check boxes and select the preferences for alignment or font sizes. In addition, groundwater labelling and the width of the graphical visualisation of any grouted section can be edited.

8.4.10 "Graph grid preferences" menu item

You can define graph grids for moments, shear forces and normal forces.

8.4.11 "Dimension lines" menu item

You can define a vertical and/or horizontal dimension line for the graphics in order to emphasise and clarify the system dimensions. If the dimension lines are displayed on the screen, the following dialog box can also be accessed directly by double-clicking the dimension line.



The distance to the retaining wall is defined by means of the "**y position**" for the horizontal dimension line and "**x position**" for the vertical dimension line. Negative values define a position above or to the left of the retaining wall. All values are in metres in the scale selected (see the menu item "**Page size + margins/Manual resize (editor)**" in Section 8.8.3).

> The fastest way to modify the position of a dimension line is to press the [**F11**] function key and then to pull the dimension line to the new position with the left mouse button pressed.

8.4.12 "Display system" menu item

Once a system has been analysed, all the state variables are automatically shown on screen. So as not to overburden the screen, certain elements of the system (for example, surcharges) are no longer shown. If you want to view all the system data without state variables, clicking this menu item will enable you to do so.

8.4.13 "Display results" menu item

After a system has been analysed, all state variables are automatically presented on the screen. If you used the menu item "**System/Display system**" to return to the system visualisation, you can go to this menu item to return to the result presentation without renewed analysis. Of course, this only works if the system has already been analysed.

8.5.1 General note

All the following information regarding the analysis results can be sent to the printer or to a file by going to the menu items "**File/Print and export**" or "**File/Print output table**". However, it is always more desirable to be able to check over the analysis results without wasting a lot of paper. The following menu items were included in the program for just this reason.

8.5.2 "Earth pressure redistribution" menu item

Here you are provided with information relating to earth pressure redistribution. This can be particularly interesting if you have calculated according to the EAB, in which case it is probably a good idea to have a copy of the EAB at hand.

8.5.3 "Main output summary" menu item

A message box appears containing the main system analysis parameters. This function can also be initialized pressing the [**F6**] function key.

You are also informed that double-clicking the left mouse button (after you have closed the message box) on any part of the graphics will cause the corresponding state variables to be shown on screen.

8.5.4 "Maximum reaction summary" menu item

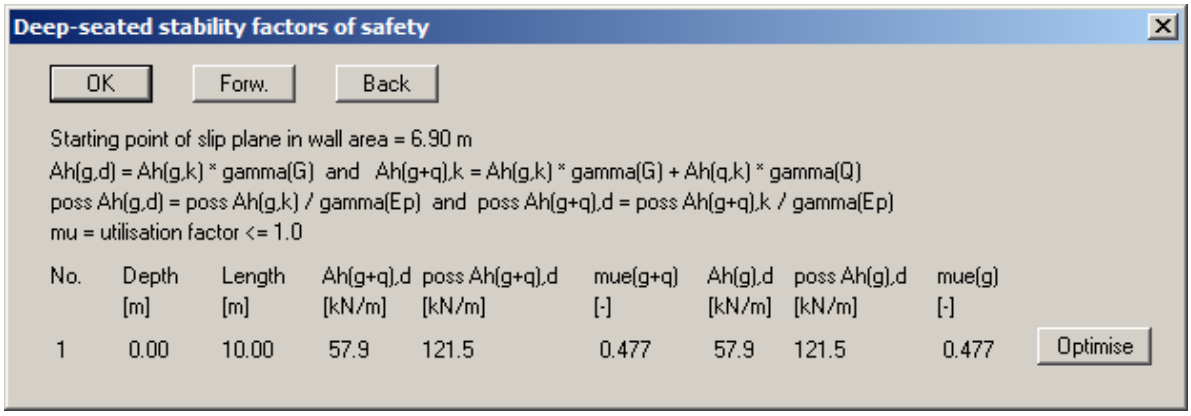
You will see the maximum values for all state variables displayed in a message box.

8.5.5 "Anchor and strut summary" menu item

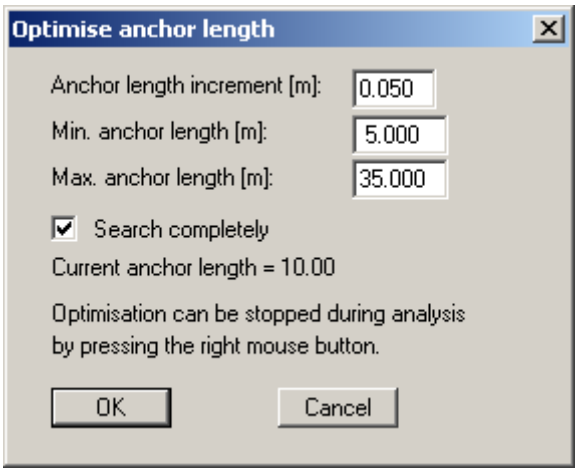
Here the design values for all the system's anchors and struts are shown in a message box.

8.5.6 "Deep-seated stability summary" menu item

After the state variables have been calculated by the program, and depending on the safety factor concept selected, the deep-seated stability safety factor or the utilisation factor of any anchors is automatically determined.



In addition to the information with regard to the anchors, the critical safety factor or the utilisation factor for deep-seated stability for each anchor is given. In the example shown here, the anchor has a safety factor which is considerably higher than the required value of 1.5. Select the "**Optimise**" button to optimise the length of the anchor.



By activating the button "**Search completely**" the optimum is searched for between the minimum and the maximum anchor length (***recommended setting!***). Otherwise, an optimum is searched for based on the current neighbouring anchor lengths. Because interaction occurs between anchors if more than one anchor is involved, it is possible for other anchors to fall below the required safety factor during the optimisation. These anchors must then be ***subsequently optimised***.

8.5.7 "Sum V FOS summary" menu item

Here you obtain all data relating to analysis of sum V displayed in a message box (see Section 7.31).

8.5.8 "Sum H FOS summary" menu item

When working on a ***soldier pile wall***, this is where you obtain all data relating to analysis of sum H displayed in a message box (see Section 7.30).

8.5.9 "Hydraulic heave FOS summary" menu item

When working with ***sheet pile walls*** (***in-situ concrete walls***) you obtain all data relating to analysis of hydraulic heave safety displayed in a message box (also see Section 7.28).

8.5.10 "Buoyancy FOS summary" menu item

When working on ***sheet pile walls*** (***in-situ concrete walls***), you obtain all data relating to analysis of the buoyancy safety displayed in a message box (see Section 7.29).

8.5.11 "Heave of anchor soil" menu item

If a value unequal to '**0**' is entered for the height of the dead man when entering the anchors in the dialog box associated with the "**Editor 2/Anchors**" menu item (see Section 8.3.7), the program performs verification of anchor soil heave similar to the method described in Section 7.3.4 of the Piling Manual.

This menu item displays the results in a message box. Verification must be activated, of course, (see menu item "**Editor 1/Deep-seated stability/heave of anchor soil**", Section 8.2.15).

8.5.12 "Heave FOS summary" menu item

When analysing the system using ***global safety factors*** the base heave safety factor is determined automatically (also see Section 7.26). If this cannot be verified you will see a corresponding warning even before analysis is complete. Otherwise, you can query the safety factor using this menu item.

8.5.13 "Verification of pull-out resistance" menu item

If analysis was performed using anchors with activated "**Verification with qs,k**" check box and verification is activated in the "**Editor 1/Pull-out resistance**" menu item (see Section 8.2.17), this menu item displays the analysis results in a message box.

8.6.1 General notes

When the system you are working on consists of a number of construction phases, each phase must initially be calculated separately and the results saved in corresponding files (see Section 8.1.4). Subsequently, from this menu, you can combine the different file contents. The following graphics can be created:

- envelope of moments;
- envelope of shear forces;
- envelope of normal forces;
- sum of displacements as a new bending line.

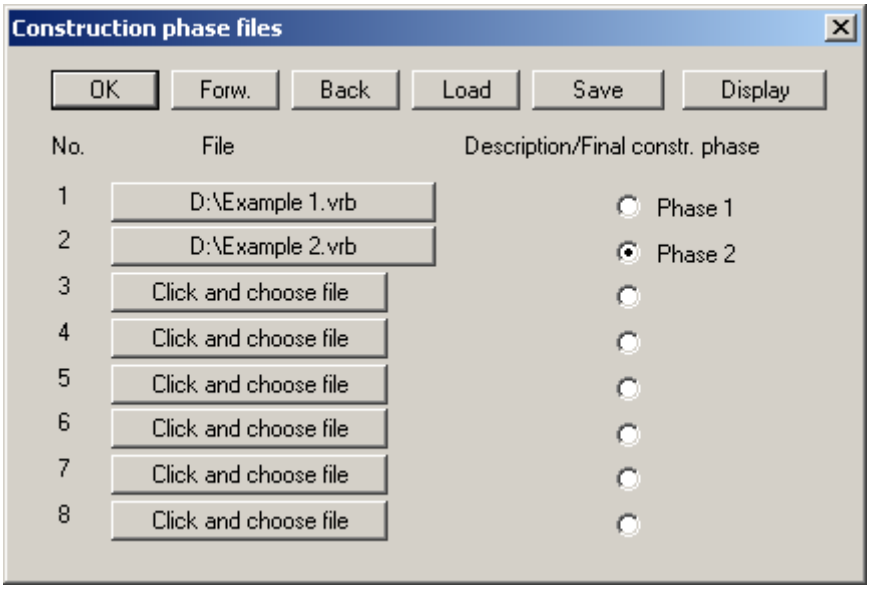
The resulting graphic is displayed in the current system. The "**Construction phases**" menu will only work with the results from the files compiled using this menu.

8.6.2 "Info" menu item

You will see information on the options provided by the "**Construction phases**" menu.

8.6.3 "Select files" menu item

Using this dialog box you can compile a list of files to be summarised. A total of 15 files can be loaded into the list. The files must be saved together with the analysis result data, otherwise an error message appears.



The current system results are not displayed automatically. They must first be saved as a result file and then be loaded into the file list. The currently loaded system serves only to represent the excavation, which is displayed in the graphics in addition to the summarised results. You can move through the list using "**Forw.**" and "**Back**". Using the "**Save**" button the selected files can be saved in order to be loaded into the application at a later session. If you save the current system, the file names will also be saved in the dataset. Using the "**Display**" button, the summarised files will be displayed on the screen.

After clicking one of the "**Click and choose file**" buttons, the desired file can be selected. The button is then labelled with the file name. The project identification is shown after this (in the example above "**Phase 1**", "**Phase 2**"). If you need to remove a file from the list, click on the button with the file name and select the "**Cancel**" button from the file selector box. The file sequence should correspond to the construction phase sequence. The radio button behind the file name serves to indicate up to which file the summary should be displayed.

There are two arrow icons  in the toolbar (see Section 8.7.6). These allow to easily switch between the individual construction phases, facilitating an animation-like visualisation of the bending line in the individual construction phases. Of course, the arrow icons only work if files have been selected and you have left the dialog box using the "**Display**" button.

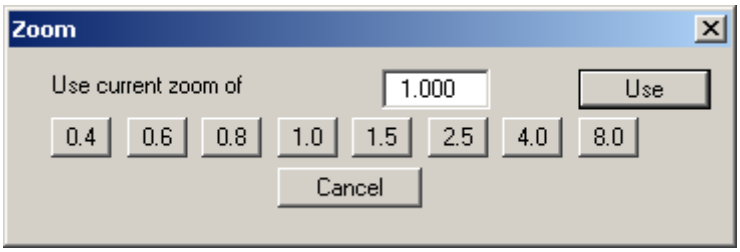
8.6.4 "Display summary" menu item

If the system graphics or the results of the current analysis are actually displayed on the screen, this option will switch to a visualisation of the construction phases.

8.7.1 "Refresh and zoom" menu item

The program works on the principle of ***What you see is what you get***. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the screen is not refreshed after every alteration.

If, e.g., after using the zoom function (see below), only part of the image is visible, you can achieve a complete view using this menu item.



A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "**Use**" to exit the box the current factor is accepted. By clicking on the "**0.4**", "**0.6**", etc. buttons, the selected factor is used directly and the dialog box closed.

It is much simpler, however, to get a complete overview using [**Esc**]. Pressing [**Esc**] allows a complete screen presentation using the zoom factor specified in this menu item. The [**F2**] key allows screen refreshing without altering the coordinates and zoom factor.

8.7.2 "Zoom info" menu item

By clicking two diametrically opposed points you can enlarge a section of the screen in order to view details better. An information box provides information on activating the zoom function and on available options.

8.7.3 "Legend font selection" menu item

With this menu item you can switch to a different true-type font. All available true-type fonts are displayed in the dialog box.

8.7.4 "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for various graphic elements (e.g. moment, shear force, loads, etc.). You can edit the pen widths for the elements shown in the dialog box; by clicking on the button with the element designation you can also edit the pen or fill colours.

On ***monochrome printers*** (e.g. laser printers), colours are shown in a corresponding grey scale. Graphic elements employing very light colours may be difficult to see. In such cases it makes sense to edit the colour preferences.

8.7.5 "Mini-CAD toolbar" and "Header toolbar" menu items

Using these two menu items you can add free text to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.). The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the "**Mini-CAD**" manual provided. The differences between the Mini-CAD and Header CAD are as follows:

- Objects created with "**Mini-CAD**" are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any foundations).
- Objects created with the "**Header CAD**" are based on the page format (in mm). This makes you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see "**Mini-CAD**" user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in exactly the same position on the page, which greatly simplifies the creation of general page information.

8.7.6 "Toolbar preferences" menu item

After starting the program a horizontal toolbar for menu items appears below the program menu bar. If you would rather work with a popup window with several columns, you can specify your preferences using this menu item. The smart icons can also be switched off.

At the bottom of the program window you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-RETAIN.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next time the program is started.

By clicking on the tools (smart icons) for the menu items you can directly reach most of the program functions. The meaning of the smart icons appears as a text box if you hover with the mouse pointer over the tools. Some of the tool functions cannot be activated from the normal menu items.

 "**Next page**"/"**Previous page**"
Using this icon, you can navigate between the individual pages in the ***tabular representation***.

 "**Select page**"
If you are in the ***tabular representation***, you can use this icon to jump to a specific page or to return to the ***normal representation***, that is, to the graphics.

 "**Zoom out**"
If you have previously ***zoomed in***, this tool returns to a full screen display.

 "**Zoom (-)**"/"**Zoom (+)**"
With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.

 **"Colour/hatching"**
Four colour settings are possible; you can click through them in sequence. A colour filled representation is the default setting, the next click shows the hatching, then coloured and hatched. The fourth click shows a representation with neither colour nor hatching. The next click starts again.

 **"Copy/print area"**
Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**, Section 9.4).

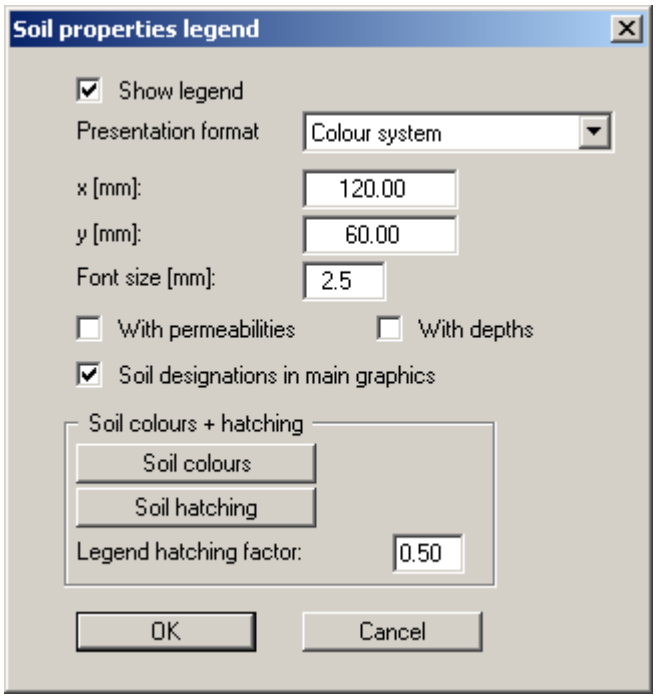
 **"Construction phase back"/"Construction phase forwards"**
If you have activated a summarised result of various construction phases, you can move from phase to phase using the arrow buttons.

 **"Undo"**
By clicking this icon the last performed change (e.g. movement of graphical elements made using the [**F11**] function key or the menu item **"Graphics preferences/Move objects"**) can be undone.

 **"Restore "**
By clicking this symbol, the last change undo carried out using **"Undo"** can be restored.

8.7.7 "Soil properties legend" menu item

A legend containing the soil properties is displayed with the graphics. If the **"Show legend"** check box is activated you can edit the representation in the dialog box for this menu item.



You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**". The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position while holding the left mouse button.

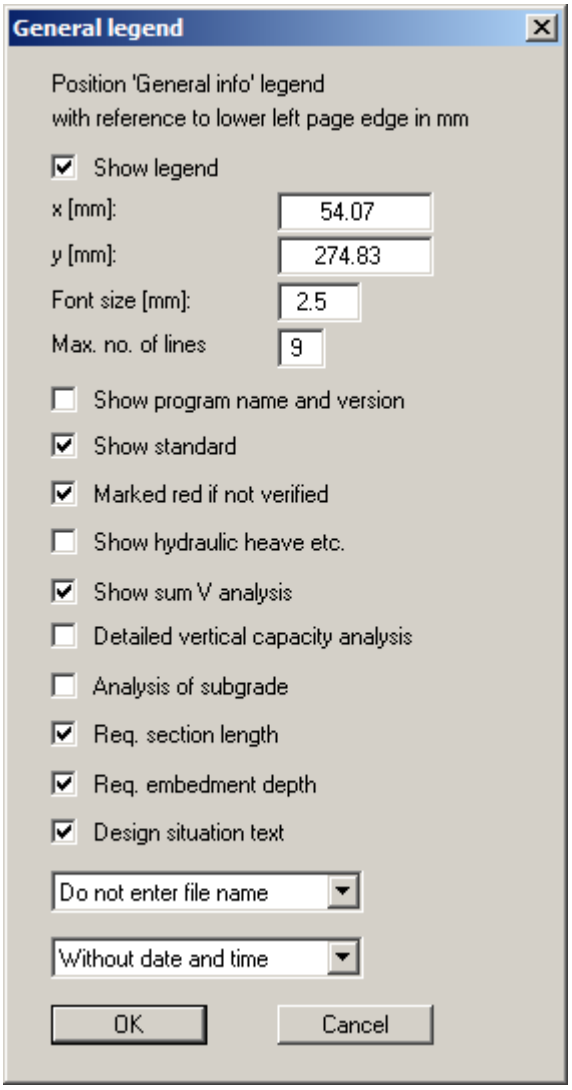
By activating the "**With permeabilities**" and "**With depths**" check boxes the corresponding data of the individual soil layers will be shown in the soil properties legend. The designationsentered for the individual ground layers are displayed in the system graphics adjacent to the layer depths if the "**Soil designations in main graphics**" check box is activated.

If "**System coloured**" is activated as presentation format in the combi-box, the soils will be displayed coloured both in the soil properties legend and in the system graphics. You can also define either hatching or colour fill and hatching for the different soil types in the combi-box. The various preferences can also be accessed via the "**Colour/hatching**" icon in the menu items toolbar (see Section 8.7.6). If you select "**System without all**", the soils are merely numbered. The required settings can be made in the "**Soil colours + hatching**" group box:

- "**Soil colours**"
  You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the "**Soil colours/Reorganise**" command button. You can save your colour preferences to a file with "**Soil colours/Save**" and use them for different systems by means of the "**Soil colours/Load**" command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.
- "**Hatching**"
  Opens a dialog box in which you can define different hatching for each soil.
- "**Legend hatching factor**"
  Input here allows tighter hatching in the soil legend. Input < 1.00 can be useful if the hatching spacing is so large that the differences in the hatching of individual soils can no longer be properly discerned in the relatively small boxes used in the legend.

8.7.8 "General legend" menu item

A legend with general properties will be displayed on your output sheet if you have activated the "**Show legend**" check box. Using this menu item you can alter the type of visualisation.

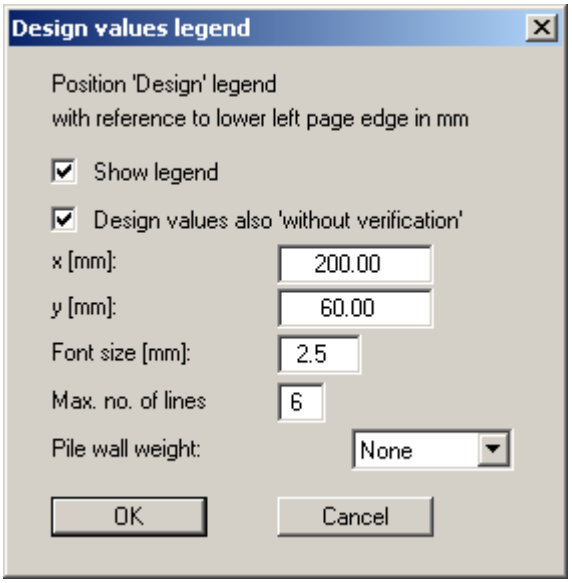


You can define and edit the position of the legend using the values "**x value**" and "**y value**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used. The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

In the ***General legend*** you can, if wished, display information on the program (name and version), on the adopted standard and design situation (load case) as well as on the current file (name, path, time info). Additionally the display of further information as e.g. "**Show sum V analysis**" can be activated depending on the selected retaining wall. Any project identification entered in the "**File/New**" or "**Editor 1/Analysis options**" dialog box will be shown automatically in the general legend.

8.7.9 "Design legend" menu item

Following analysis and design a legend containing the principal system design results is displayed on the screen. Using this menu item you can alter the type of presentation if the "**Show legend**" check box is activated.



You can define and edit the position of the legend using the values "**x value**" and "**y value**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used. The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

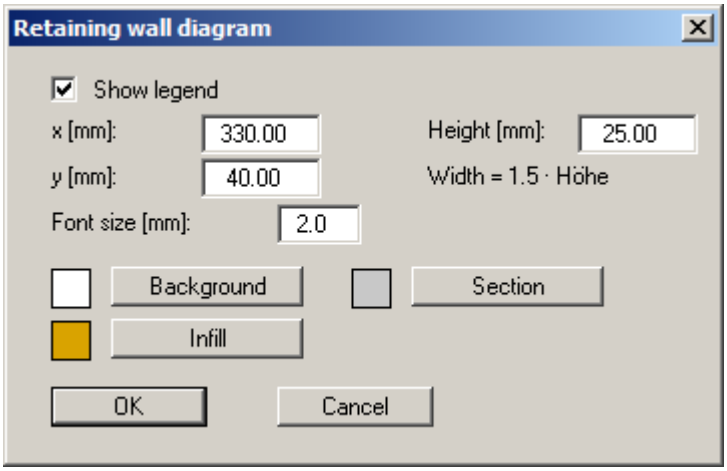
8.7.10 "Subgrade modulus legend" menu item

A legend is displayed on the screen containing the subgrade reaction moduli entered for the sections along the retaining wall defined for the subgrade reaction profile. Using this menu item you can alter the type of presentation if the "**Show legend**" check box is activated.

You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**". The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position while holding the left mouse button.

8.7.11 "Retaining wall diagram" menu item

A legend containing a small sketch of the retaining wall employed is displayed on the screen. Using this menu item you can alter the type of presentation or turn off the legend completely. The following dialog box opens for a ***soldier pile wall***, for example.



> If you edit the font sizes you may have to alter the size of the legend to suit, otherwise the dimensions in the legend may no longer be visible.

8.7.12 "Move objects" menu item

Select this menu item in order to position legends, diagrams and other graphical elements at the desired position on the output sheet. You can also move objects by pressing [**F11**] and then positioning the legend box with the left mouse button pressed. In that case an info-box appears no more.

If "**Manual graph positioning**" has been selected in the "**System/Graph positioning preferences**" menu item (see Section 8.4.7), the result graphs can also be repositioned using this function.

8.7.13 "Save graphics preferences" menu item

Some of the preferences you made with the menu items of the "**Graphics preferences**" menu can be saved to a file. If you select "**GGU-RETAIN.alg**" as file name, and save the file on the same level as the program, the data will be automatically loaded the next time the program is started and need not be entered again.

> If you do not go to "**File/New**" upon starting the program, but open a previously saved file instead, the preferences used at the time of saving are shown. If subsequent changes in the general preferences are to be used for existing files, these preferences must be imported using the menu item "**Graphics preferences/Load graphics preferences**".

8.7.14 "Load graphics preferences" menu item

You can reload a graphics preferences file into the program, which was saved using the "**Graphics preferences/Save graphics preferences**" menu item. Only the corresponding data will be refreshed.

8.8.1 "Auto-resize" menu item

This menu item provides a to-scale visualisation, in both x and y coordinates, of the system and result graphics. If you have previously altered the image coordinates graphically or via editor, you can quickly achieve a complete view using this menu item. This function can also be accessed using the [**F9**] function key.

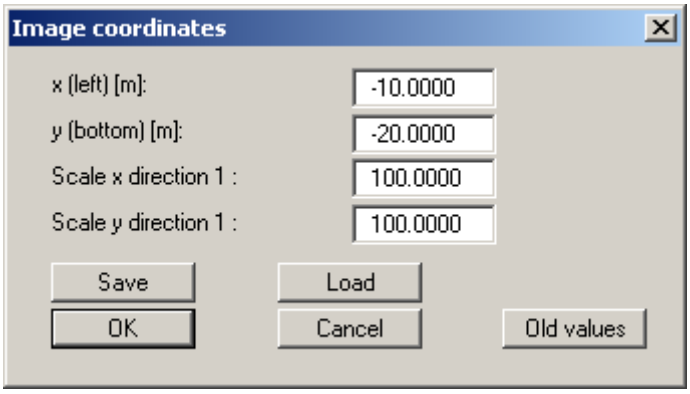
8.8.2 "Manual resize (mouse)" menu item

You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [**Ctrl**] and [**Shift**] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (scale x-direction/scale y-direction) need to be retained, the "**Proportional section**" check box must be activated.

Alternatively, you can simply "**Redefine origin**" of the visualisation. The previous scale preferences are not affected by this.

8.8.3 "Manual resize (editor)" menu item

You can alter the image coordinatesvia editor by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the ***drawing area***. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 8.8.6).



The image coordinates entered here can be saved in a file with the extension "**.bxy**" and be reloaded later for the same file or for different files.

If you want to recover the previous values during input or use the menu item again after editing the coordinates, you can do this by pressing the "**Old values**" button.

8.8.4 "Zoom" menu item

You can linearly enlarge or reduce the image coordinates.

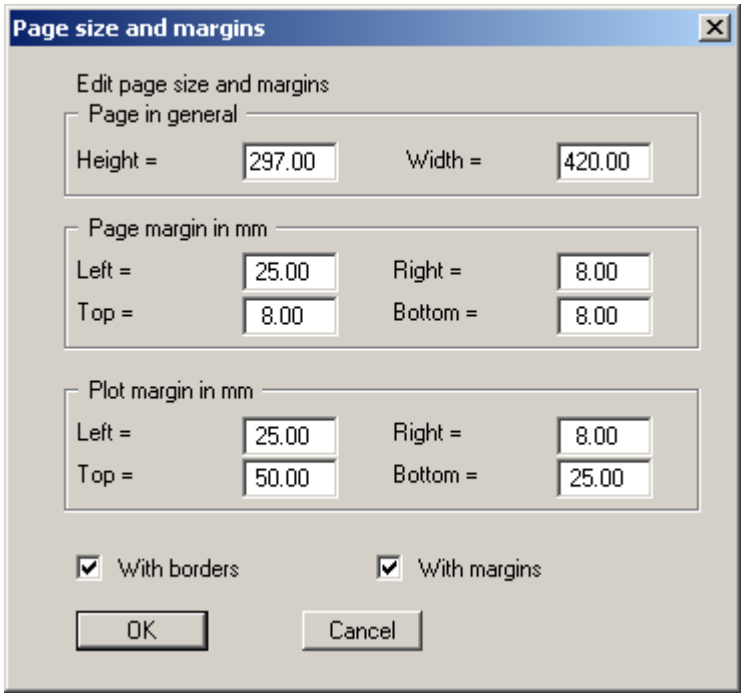
8.8.5 "Font size selection" menu item

You can edit font sizes for labelling the various drawing elements.

The font sizes of text within legends are edited in the respective legend editor. Just double-click in a legend to do this.

8.8.6 "Page size and margins" menu item

The default page set-up is A3 when the program is started. You can edit the page format in the following dialog box.



- "**Page in general**" defines the size of the output sheet. The A3 format is set as default. The program automatically draws thin cutting borders around the page, which are required when using a plotter on paper rolls. The borders can be switched off using the "**With borders**" check box.
- "**Page margin**" defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram. You can switch off the frame deactivating the "**With margins**" check box.
- The "**Plot margin**" define a set distance between the page margin and the actual ***drawing area*** in which the graphical evaluation of your input is presented.

8.8.7 "Undo" menu item

If you have carried out any changes to dialog boxes or moved objects to a different position on the screen after selecting the "**Graphics preferences/Move objects**" menu item or using the [**F11**] function key, this menu item will allow you to undo the movements. This function can also be reached by using the key combination [**Alt**] + [**Back**] or the appropriate tool in the toolbar (see Section 8.7.6).

8.8.8 "Restore" menu item

When this menu item is selected the last change made in a dialog box or the last change in the position of objects, which you undid using the menu item "**Page size + margins/Undo**" will be restored. This function can also be reached by using the key combination [**Ctrl**] + [**Back**] or the appropriate tool in the toolbar (see Section 8.7.6).

8.8.9 "Preferences" menu item

You can activate or deactivate the undo functions.

8.9 ? menu

8.9.1 "Copyright" menu item

You will see a copyright message and information on the program version number.

The "**System**" button shows information on your computer configuration and the folders used by **GGU-RETAIN**.

8.9.2 "GGU on the web" menu item

Using this menu item you can access the GGU Software website: www.ggu-software.com.
Keep in touch with new program versions and the regular ***download*** offers.

If you would like to be automatically notified about program innovations, please register for the Newsletter in our Knowledge Base. Go to the following website: http://kbase.civilserve.com.

8.9.3 "GGU support" menu item

This menu item takes to the GGU-Software Support area at www.ggu-software.com.

8.9.4 "Maxima" menu item

Here you can check the defaults for maximum values.

8.9.5 "Active wall friction angle" menu item

A message box appears with information relating to active angle of wall friction according to the EAB.

8.9.6 "Compare earth pressure coefficients" menu item

Here you can calculate the earth pressure coefficients for given values of phi, delta and beta.

8.9.7 "Vertical capacity" menu item

You can compare qb,k and qs,k for given qc and cu,k values to EAU 2012 and Recommendations on Piling 2012.

8.9.8 "Aktive earth pressure (constrained slip surface)" menu item

The active earth pressure is determined for a constrained slip surface.

8.9.9 "Steel design to DIN EN 1993" menu item

Design values can be entered by hand irrespective of any analysis results. They are then used to perform steel design to EN 1993/EC 3.

8.9.10 "Help" menu item

The **GGU-RETAIN** manual is opened as a PDF document. The help function can also be accessed using the [**F1**] function key.

8.9.11 "What's new?" menu item

You will see information on program improvements in comparison to older versions.

8.9.12 "Language preferences" menu item

This menu item allows you to switch the menus and graphics from German to English or Spanish and vice versa. The program always starts with the language setting applicable when it was last ended.

9 Tips and tricks

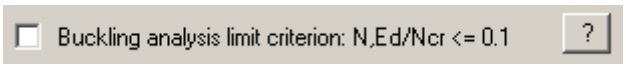
9.1 "?" buttons

Reading of the manual can mostly be dispensed with, because

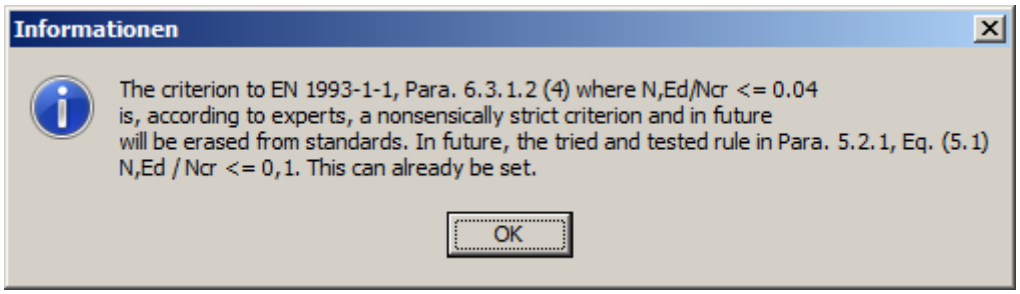
"**?**" buttons 

dealing with almost all geotechnical and program-specific problems are available in the dialog boxes. You are presented with the necessary information by clicking the "**?**" button.

For example, the following check box can be found in the "**Editor 1/Analysis options**" dialog box:

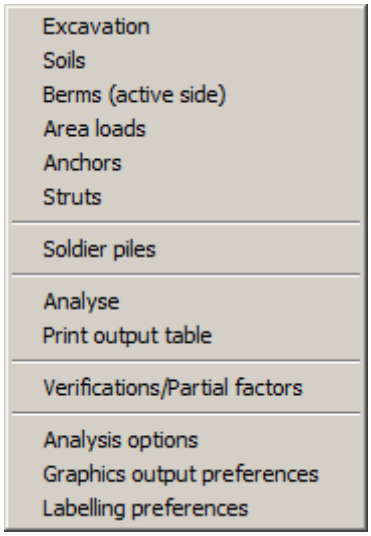


If you click on the question mark, you will see the following message box:



9.2 Keyboard and mouse

If you click the right mouse button anywhere on the screen a context menu containing the principal menu items opens.



By double-clicking the left mouse button on legends or **Mini-CAD** objects, the editor for the selected element immediately opens, allowing it to be edited.

If you double-click in the system graphics with the left mouse button you will see a message box with the results at the selected depth. This gives you a quick overview of the results for any desired depth.

You can scroll the screen with the keyboard using the cursor keys and the [**Page up**] and [**Page down**] keys. By clicking and pulling with the mouse, with [**Ctrl**] pressed, you activate the zoom function, i.e. the selected section will fill the screen. Use the mouse wheel to zoom in or out of the screen view or to pan.

In addition, scale and coordinates of the system graphics (drawing area within the plotting margins) can be altered directly using the mouse wheel. The following mouse wheel functions are available:

Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):

- [**Ctrl**] + mouse wheel up = enlarge system graphics (change of scale)
- [**Ctrl**] + mouse wheel down = shrink system graphics (change of scale)
- [**Shift**] + mouse wheel up = move system graphics up (change in system coordinates)
- [**Shift**] + mouse wheel down = move system graphics down (change in system coordinates)
- [**Shift**] + [**Ctrl**] + mouse wheel up = move system graphics right (change in system coordinates)
- [**Shift**] + [**Ctrl**] + mouse wheel down = move system graphics left (change in system coordinates)

Change screen coordinates:

- Mouse wheel up = move screen image up
- Mouse wheel down = move screen image down
- [**Alt**] + [**Ctrl**] + mouse wheel up = enlarge screen image (zoom in)
- [**Alt**] + [**Ctrl**] + mouse wheel down = shrink screen image (zoom out)
- [**Alt**] + [**Shift**] + mouse wheel up = move screen image right
- [**Alt**] + [**Shift**] + mouse wheel down = move screen image left

9.3 Function keys

Some of the function keys are assigned program functions. The allocations are noted after the corresponding menu items. The individual function key allocations are:

- [**Esc**] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [**F1**] opens the manual file.
- [**F2**] refreshes the screen without altering the current magnification.
- [**F5**] opens the menu item "**System/Analyse**".
- [**F6**] opens the menu item "**Evaluation/Main output summary**".
- [**F9**] opens the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

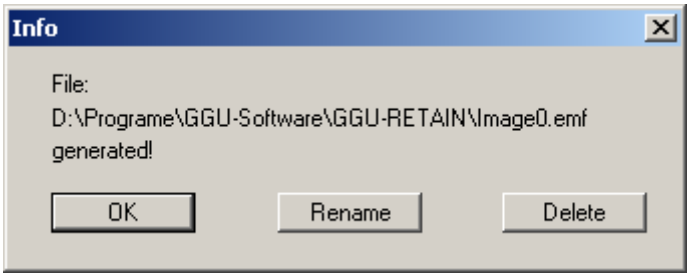
9.4 "Copy/print area" icon

A dialog box opens when the "**Copy/print area**" icon  in the menu toolbar is clicked, describing the options available for this function. For example, using this icon it is possible to either copy areas of the screen graphics and paste them into the report, or send them directly to a printer.

In the dialog box, first select where the copied area should be transferred to: "**Clipboard**", "**File**" or "**Printer**". The cursor is displayed as a cross after leaving the dialog box and, keeping the left mouse button pressed, the required area may be enclosed. If the marked area does not suit your requirements, abort the subsequent boxes and restart the function by clicking the icon again.

If "**Clipboard**" was selected, move to the MS Word document (for example) after marking the area and paste the copied graphics using "*Edit/Paste*".

If "**File**" was selected, the following dialog box opens once the area has been defined:



The default location of the file is the folder from which the program is started and, if several files are created, the file is given the file name "**Image0.emf**" with sequential numbering. If the "**Rename**" button in the dialog box is clicked, a file selector box opens and the copied area can be saved under a different name in a user-defined folder. Saving can be aborted by pressing the "**Delete**" button.

If the "**Printer**" button was pressed in the first dialog box, a dialog box for defining the printer settings opens after marking the area. Following this, a dialog box for defining the image output settings opens. After confirming the settings the defined area is output to the selected printer.

10 Index

2

A

B

C

D

E

F

G

H

I

K

L

M

N

O

P

R

S

T

U

V

W

Y

Z